DECO GEOPHYSICAL
software company

RadExPro Plus v. 2011.4
User Manual

(revised 26.01.2012)

DECO Geophysical

Software Company

MSU Science Park,
Leninskie Gory 1-77
119992 Moscow, Russia
Tel. (+7 495) 930 8414
Fax (+7 495) 930 8058
www.radexpro.com

E-mail:
support@radexpro .ru
sales@radexpro.ru

Table of Contents

Introduction....6
System Requirements....8
Contact Information....9
Installation....10
Working With the Software....11
Project Management....14
What is a project?....14
Creating a new project....15
Opening an existing project....20
Working with Projects....21
Starting work with a project....21
Working with project trees....27
Working with flows....37
Flow construction (simple mode)....38
Complex (graphical) mode of flow creation....41
Switching between flow editing modes....44
Flow execution....44
Logging....45
Adding data files to the project....48
Adding a data file....48
Database Object Selection window - saving and opening project database objects....53
Geometry Spreadsheet - spreadsheet editor for trace header values....58
Using the geometry spreadsheet editor....58
Using the editor of mathematic operations with headers....61
Import of ASCII files....61
Trace Enumerator tool....68
Import of SPS-X files....69
UKOOA p1-90 files import....72
RadExPro header fields....76
Edit Header Fields....76
Important header fields....78
Processing modules....80
Data I/O (Data input/output)....80
Trace input....80
Trace Output....81
SEG-Y input....82
SEG-Y output....86
SEG-D Input....90
SEG-B Input....95
SEG-2 Input....96
Text Output....98
SCS-3 Input....99
Super Gather....101
Data Input....102
Data output....106
RAMAC/GPR....107
LOGIS....108
GSSI....108

Geometry/Headers....109
Trace Header Math....109
Near-Surface Geometry Input....114
Header Averager....119
Shift header....120
Header<->Dataset Transfer....120
Surface-Consistent Calibration*....122
Compute Line Length....123
Interactive Tools....124
Screen Display....124
Plotting*....150
3D Gazer (Display of seismic crossections in real 3D coordinates)....165
Signal Processing....179
Amplitude Correction....179
DC Removal....181
Bandpass Filtering....181
Butterworth Filtering....184
Resample....186
Hilbert Transforms....186
Trace Math Transforms....187
Data Enhancement....189
2D Spatial Filtering....189
Antenna Ringdown Removal....191
Radial Trace Transform....191
Radon Transforms....196
Burst Noise Removal....200
Wave field subtraction....201
F-X Deconvolution....210
Spectral whitening....212
Spectral shaping....214
F-K Filter....216
Trace Editing....223
Trace editing....223
Trace length (Trace length changing)....225
Trace Math....226
Deconvolution....228
Deconvolution (Deterministic deconvolution)....228
Custom Impulse Trace Transforms....231
Predictive Deconvolution....234
Surface-Consistent Deconvolution....235
Nonstationary predictive deconvolution....240
Static Corrections....246
Calculate Statics....246
Apply Statics....253
Auto Statics* (Automatic static correction calculation....255
Picking (It is old module. We recommend you to use Statics Correction module)....264
Auto Statics* (Calculation of automatic statics)....274
Velocity....281
Time/Depth Conversion....281
DB Velocity Interpolation....285

HVA Semblance (Horizon Velocity Analysis Semblance)....286
NMO/NMI....289
HVA (Horizontal Velocity Analysis)....292
Velocity Editor....307
Interactive Velocity Analysis....311
Velocity Curve Editor....320
Stacking/Ensembles....324
Ensemble Stack....324
Migration....326
Stolt F-K Migration....326
STOLT3D (3D Stolt F-K Migration)....327
T-K Migration....328
VSP....330
VSP Migration....330
Curved Profile VSP Migration....330
2D-3D VSP Migration....330
3C Orientation....344
2C Rotation....346
sym2ort....347
VSP Geometry*....348
VSP Data Modeling....354
VSP SDC (Vertical Seismic Profiling Spherical Divergence Correction for non-converted reflected compressional waves)....357
Advanced VSP Display....360
VSP NMO....371
QC (quality control)....373
Ensemble QC....373
Correlation function compute....377
Compute fold....379
Apparent velocity calculation....380
SSAA (Seismic Sequence Attribute Analysis)....381
CrossPlot (Tied crossplots)....385
3C Processing (multi-component processing)....416
Asymptotic CCP Binning....416
Modeling....420
Hodograph....420
Auto Picking....423
First Breaks Picking....423
Marine....424
Zero-Offset DeMultiple....424
Tides Import*....429
Marine Geometry Input....431
Deghosting* (Ghost wave suppression on near offsets)....435
Interpolation....436
Profile Interpolation* (Interpolation of profile data on a regular grid)....436
CCP-CMP X Interpolation....442
Refraction....444
Travel Time Inversion*....444
Easy Refraction* (Processing of seismic refraction data)....461
Surface Wave Analysis....477

MASW (Multichannel Analysis of Surface Waves)....477
Special tools....504
Map....504
The menu and tool bar....504
Status Bar....511
Keyboard and commands....511
3D CDP Binning....513
Working in binning mode....515
Toolbar....516
Sequence of work with the application....517
Tape Loader (program of reading data from tapes)....518
System requirements....518
Working with the program....518
Well loading and displaying in the RadExPro package....521
Wells loading to database....521
Well displaying....525
File formats....528
Refraction mode....530
Development tools - Creation of your own module....531
Microsoft Visual C++ Custom AppWizard....531
FUNCTIONS....533

Introduction

The **RadExPro** software is dedicated to comprehensive processing and interpretation of mainly near-surface on-shore and off-shore 2D seismic, VSP and GPR data, as well as to 2D/3D field Quality Control (QC).

The software operates on standard Windows-driven PCs and does not make any distinctive demands on specific hardware or require any additional software.

The scope of the system includes:

- Processing and interpretation of 2D on-shore and marine seismic data.
- Field QC of 2D/3D seismic data.
- Refraction seismic data processing and interpretation.
- MASW
- VSP data processing.
- GPR data processing and interpretation.

The **RadExPro** is available in several configurations united by a common user-friendly graphic interface but different in the set of embedded processing modules:

- **Basic** - basic processing of near-surface reflection, refraction seismic data processing, analysis of surface waves (MASW) ;
- **Standard** - basic processing of near-surface reflection, refraction seismic data processing, analysis of surface waves (MASW), vertical seismic profile (VSP) data processing;

- **QC** - field processing and QC of 2D/3D seismic data, advanced processing of near-surface reflection data, processing of refraction seismic data, analysis of surface waves (MASW);

- **Advanced** - field processing and QC of 2D/3D seismic data, advanced processing of near-surface reflection data, processing of refraction seismic data, analysis of surface waves (MASW), processing of VSP data.

This manual describes the **RadExPro Advanced** configuration. If you have one of the other configurations of the software, be aware that some features described in this manual are not present in your configuration. You can check the content of different configurations on our Web-site: http://www.radexpro.com

System Requirements

Minimum:

- Pentium4 CPU
- 256 Mb RAM
- OS MS Windows 98/Me/NT4/2000/XP/Vista
- SVGA monitor

Recommended minimum:

- Intel Core 2 Duo CPU
- 1 Gb RAM
- OS MS Windows XP/2000/Vista/7
- 1280x1024 monitor

Contact Information

Any questions and requests regarding functions of the software may be addressed to the ***DECO Geophysical Software Company***:

DECO Geophysical SC

Moscow State Univeristy Science Park

Leninskie Gory 1-77

119992 Moscow

Russia

Tel.(+7 495) 930 84 14

Fax.(+7 495) 930 80 58

Internet: www.radexpro.com

E-mail: support@radexpro.ru

Installation

Run the software installation executable file and follow the instructions of the Installation Wizard.

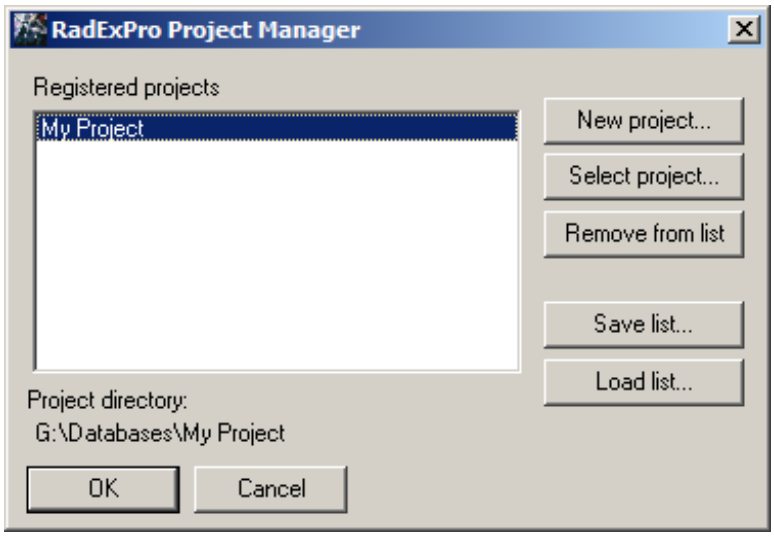
Working With the Software

There are two ways to launch **RadExPro:**

- From the **Start Menu.** The program is located in the **All Programs** menu in **DECO Geophysical** group.

- From **"Windows Explorer", "My Computer",** or any other file manager, navigate to the directory where the program has been installed and launch the file **starter.exe**.

Before the program is launched the following window will appear:



The window contains a list of already registered **RadExPro** projects and project management buttons.

- **New project...** allows creation of a new, empty project, the name of which will appear in the list of registered projects **(Registered projects)**

- **Select project...** allows adding an already existing project into the list of registered projects **(Registered projects)**

- **Remove from list** button removes the selected project from the registered projects list **(Registered projects)**

- **Save list...** button saves the list of registered projects into the text file.

- **Load list...** button loads the list of registered projects from a text file.

When required project is selected, click the **OK** button. To leave without saving changes, click **Cancel.**

Project Management

What is a project?

All data processing should be performed within the framework of a **processing project**. A project is a set of different types of data, their geometry, as well as processing flows applied to the data. Each project is kept in a separate directory on a hard disk.

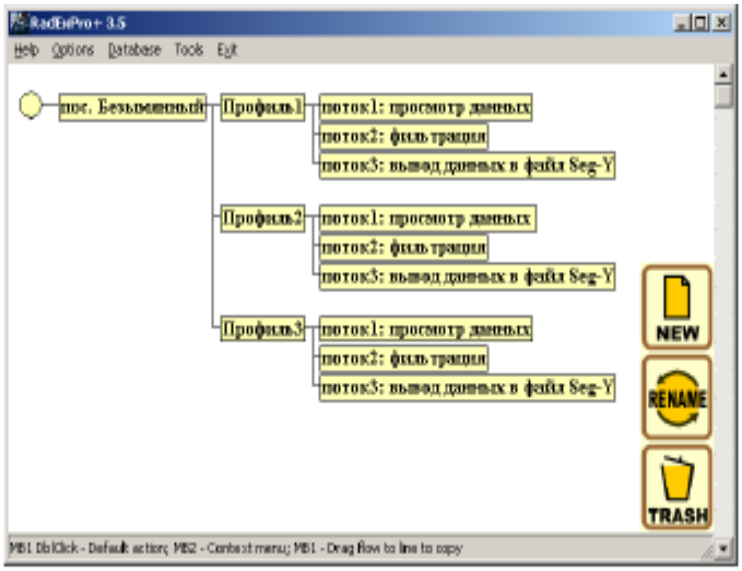
A **RadExPro** project represents a database, divided into **Area**, **Line**, and **Flow**. Area consists of one or more lines, each line, in turn, is made up of flows.

The objects of a project's data can contain seismic data, borehole data (wells), time picks (picks), velocity picks, grids and bitmap images. Each data object of the project is associated with this or that level of the database, attached to a specific area, or a specific line, or a specific flow. Although **RadExPro** provides for working with external data files, in most cases work with a data file should start from registration in the project database.

For example, you perform the processing of the data, acquired near illage along 3 lines. You may create a new project (see the section "Creation of new project"), add an area **"Unnamed"**, and inside the area you can create 3 lines: **"Profile1"**, **"Profile2"**, and **"Profile3"** (see the chapter "Working with a project", in the paragraph "Working with a project tree").

Next, you have to load data into the project. In the given case the data represent the set of field seismograms for each line (the data loading is examined in the chapter "Working with a project", in the paragraph "Registration of the data files in the project"). Seismograms of the 1st line should be tied to the level **"Profile1"**, the 2nd line tied to the level **"Profile2"** etc.

After loading the data, you may create several processing flows for each line (see the chapter “Working with a project”, in the paragraph “Working with flows”) and run the tasks.



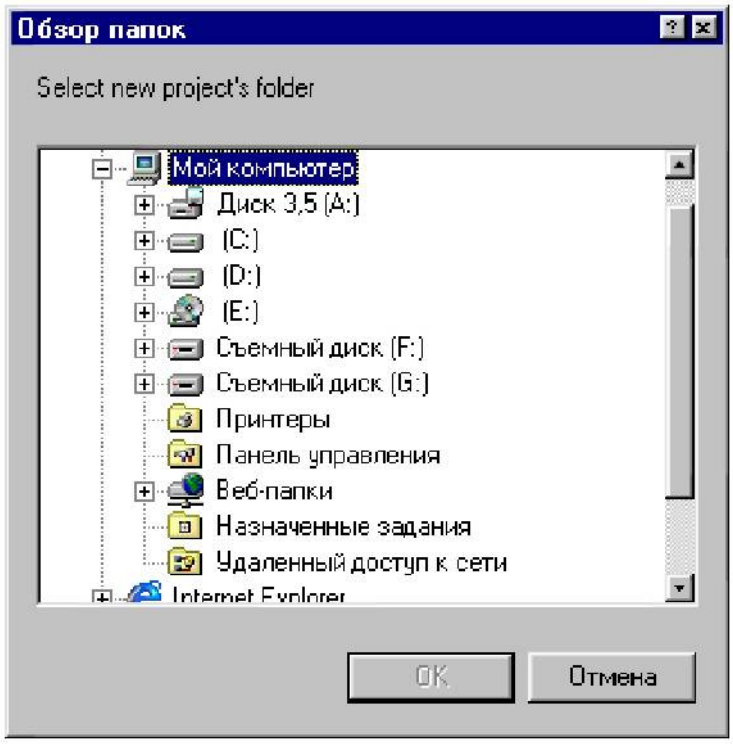
You may need to save intermediate processing results. You can do this by creating new datasets inside the project, linked to any level of a project.

You may exit the program and during the next run you can open a project created earlier with **area**, **lines**, and attached seismograms and **processing flows** (see the paragraph “Opening an existing project”). You can modify the project, add new lines or flows, change the parameters of procedures in the flows created earlier etc. All your work is well organized and stored in one place. You can look through the project to find out what had been done earlier and add something if you need to do so. After you have finished the processing, you may need to extract the processing results from the **RadExPro** project and save them as a file to a disk in one of the standard formats. There are special modules for data output to external files (see the chapter “processing modules”, in the paragraph “Data I/O”).

Creating a new project

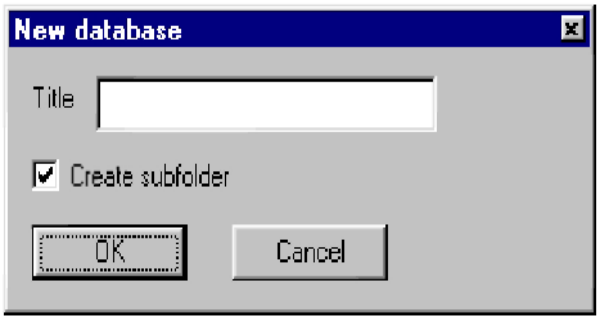
The process of new project creation starts with determination of a directory on disk, where all data objects of the project as well as its database files will be stored.

In the **RadExPro Project Manager** select a **New project...** button. A dialog window for project folder selection will appear.



Select a directory and click the **OK** button. If the **OK** button is not active, it means that the selected folder already contains another **RadExPro** project.

After the directory for a new project has been selected the **New database** window will appear. Enter the name of the new project in the **Title** field. If the **Create subfolder** option is activated, the subdirectory with the name specified in the **Title** field will be created in the selected directory. Otherwise, a new project will be created directly in the selected directory.



After all parameters have been specified, click the **OK** button. A new project will be created in the specified directory. In this case the **RadExPro** program will create five project database files in the directory:

data.fbl

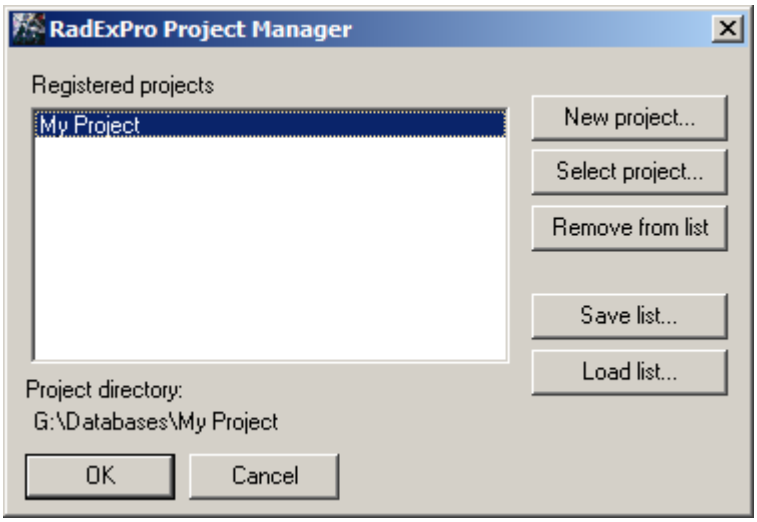
struct.fbl

data.fsp

struct.fsp

atoms.tbl

The name of a new project will appear in the registered project list **(Registered projects) of** the **RadExPro Project Manager** window.



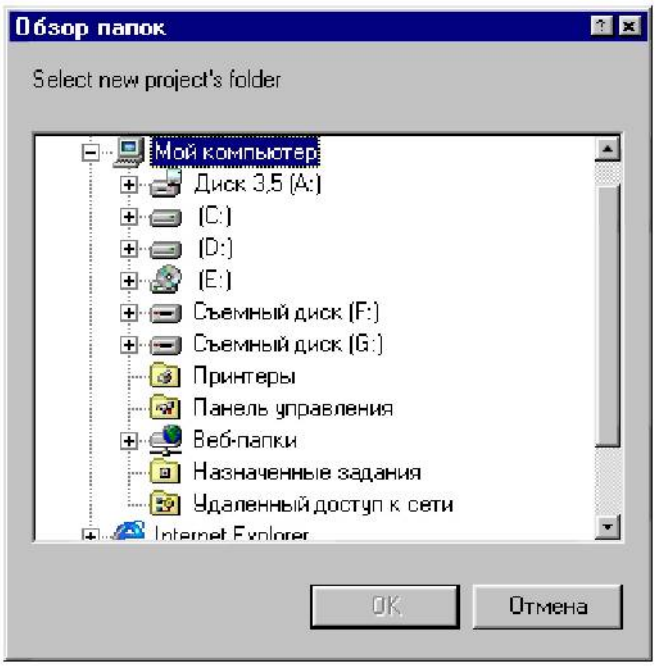
Select the created project by clicking the left mouse button **(MB1)** (in this case, in the **Project directory** field the information about the project path will be displayed) and click the **OK** button.

Opening an existing project

In order to open a project it should first be added to the registered project list **(Registered projects)** in the **RadExPro Project Manager** window.

If the required project is already on the list, you can simply select it with the left mouse button **(MB1)** (in this case, in the **Project directory** field the information about project path will be displayed) and click the **OK** button.

If the project already exists on the hard disk but its name is not displayed in the registered project list, select the **Select project...** button. The window for project directory selection will open.



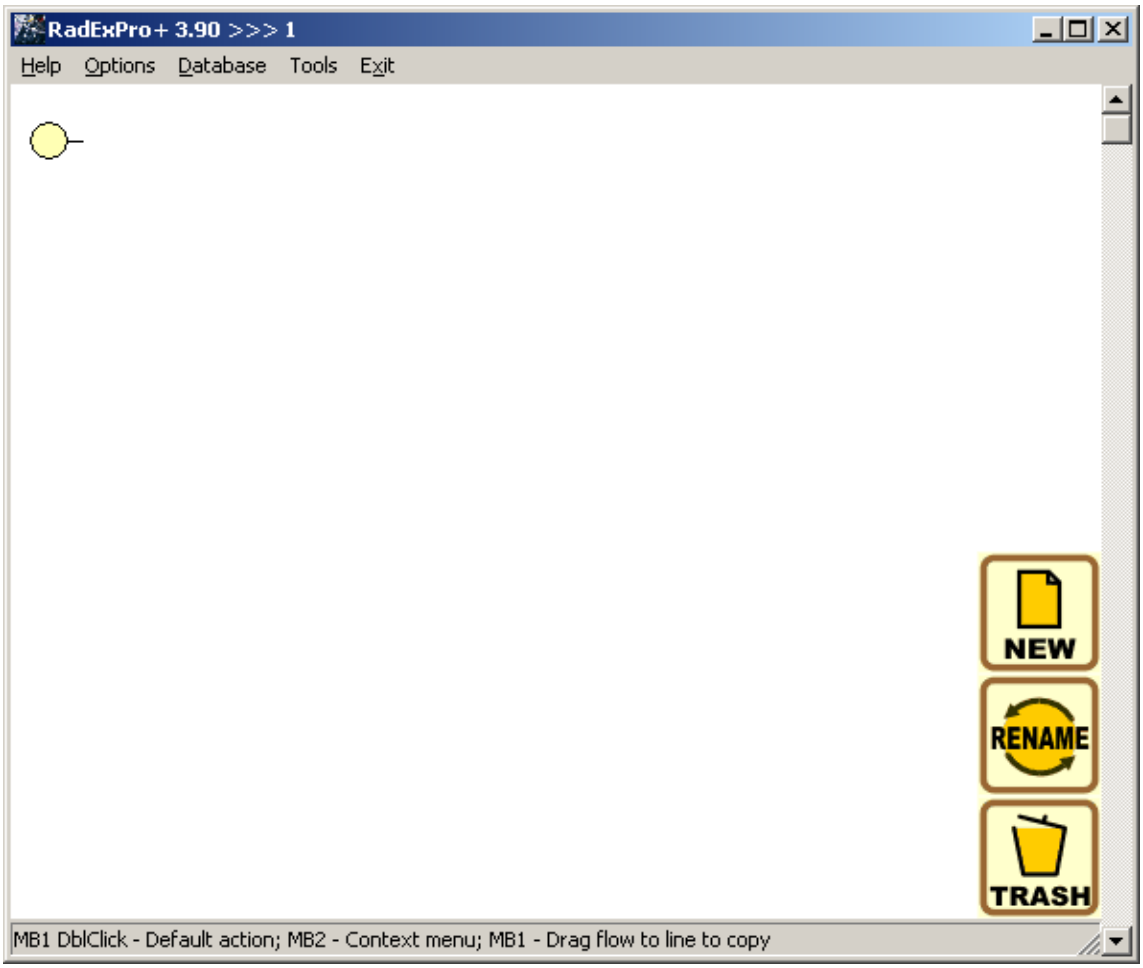
Select the directory in which the project is stored and click the **OK** button. If the **OK** button is not active, then it means that the directory selected does not contain a **RadExPro** project.

If the project directory was successfully selected then its name will appear in the registered project list **(Registered projects).** After this, the project can be opened as described at the beginning of the chapter.

Working with Projects

Starting work with a project

After a new project has been created (or an existing project has been opened) the main application window containing a graphical view of the active project tree will be displayed. If the project is a new one the window will be similar to the following:



The menu contains the following commands:

- **Help** command launches the **RadExPro** program help

- **Options** command opens the submenu containing the following commands:

Refraction mode This command switches on and off the specialized mode designed for processing of the refraction data. When this mode is on, the program respond differently to mouse

commands (see **Refraction mode** part of this manual for details).

- **Font...**. When selecting this command the dialog box for parameters selection for the font to be used for database element names will open

- **Logs...** When selecting this command, the dialog for logging parameters will open.

- **Database** command opens the submenu containing a number of commands for working with a project database: **Save** saves to the active database on disk into a special file with *.dbs extension

- **Load** loads the previously saved database from the ***.dbs-file**

- **Add data file** adds a new seismic data file to the project

- **Geometry Spreadsheet** allows you to view and edit the trace header field values

- **Database Visualization launches** the **Database Visualization** application

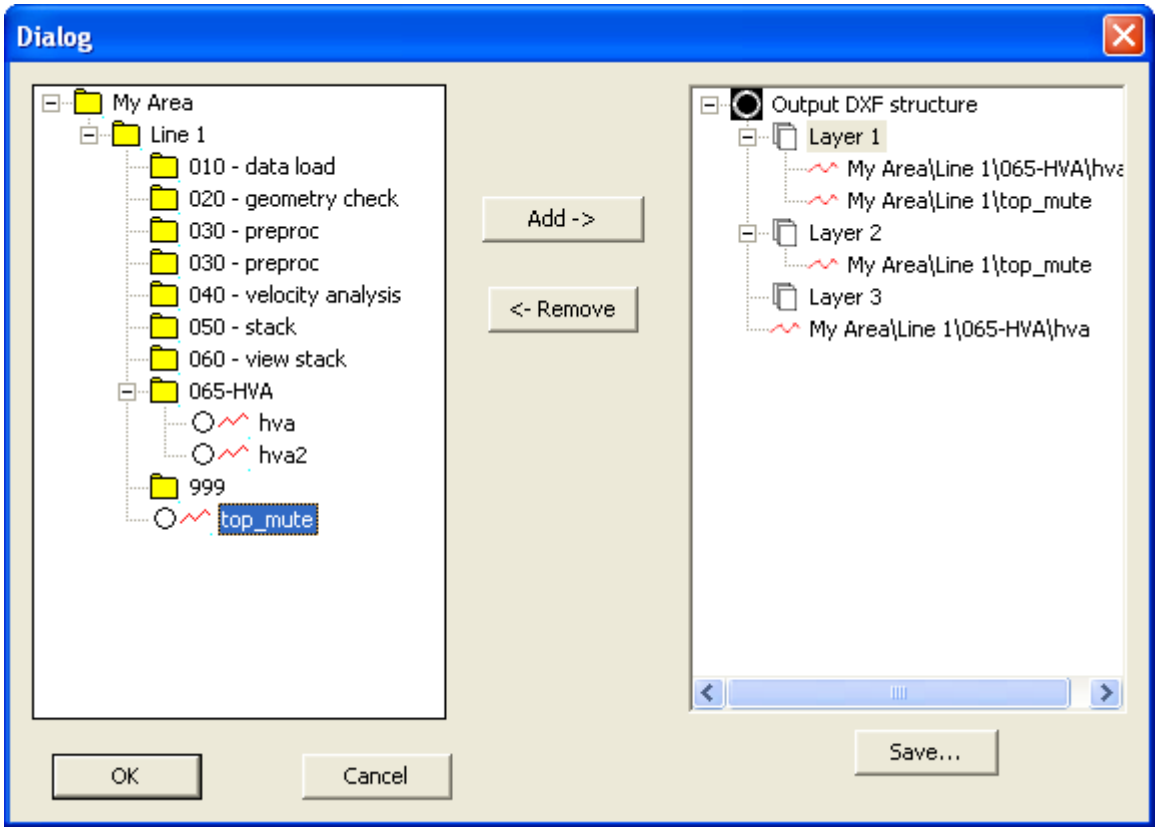
- **Database Manager** launches the **Database Manager** application

- **Database History...** shows the processing history of one or another seismic data file registered in the project

- **DXF export**

This menu entry allows saving horizon picks into the DXF format for further processing in GIS and CAD systems. When the option is selected the following dialog appears:

On the left there is a representation of the project tree with picks. On the right there is a representation of the resulting DXF file structure, possibly including layers and picks to be exported.



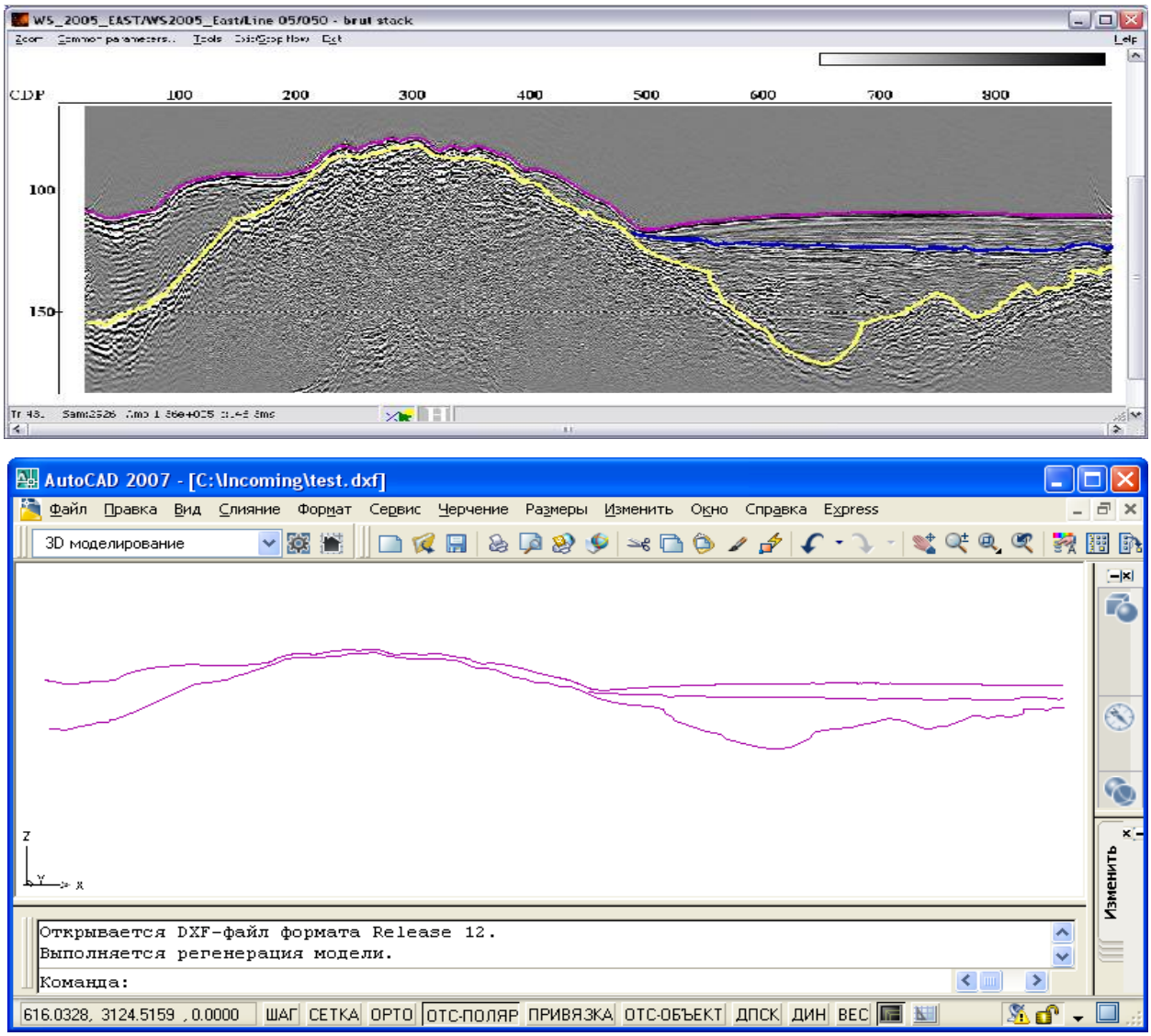
You can add picks to the DXF from the project either using the **Add** button or double-clicking the pick name. You can remove picks from the DXF by either using the **Remove** button, or through a context menu (right click on the pick name and select **Delete...** in the pop-up menu).

In the DXF you can create layers and save picks either to the root or/and to these layers. To create a layer right-click the DXF structure field and select the **New layer...** command of the pop-up menu. When created, the layer can be renamed: double-click it and type a new name.

When you are ready with populating the DXF structure with layers and picks, click the **Save...** or the **Ok** and specify the name of the output file. If the **Ok** button was used the dialog will close, while using the **Save...** button allows creating several DXF files within one session.

Coordinates. The two matching headers of the pick are interpreted as X and Y coordinates of the DXF accordingly, while the pick value is considered as Z coordinate. In case both headers are the same (e.g. CDP_X:CDP_X), the pick is considered as 2D. In this case, the header value is stored as X coordinate of the DXF while Y coordinate is assigned 0.

Below there is an example of the **RadExPro Plus** picks exported to DXF and loaded to **AutoCAD:**



- **Edit header fields** allows you to edit header fields, i.e. delete/change the existing fields and add new ones

- **Database Management** opens the submenu with two commands: **Fix data file location** and **Bulk data file location.** The first command allows you to change the paths to the source data within the database and the second command allows you to change the path to the directory containing all project files. The application of these commands may be required when loading a previously saved database structure from the ***.dbs-file**

- **Tools** command opens a submenu containing the following commands:

- **View Map** launches the **Map** application
- **Velocity model editor** launches the **Velocity model editor** application
- **Velocity Picks To CMP Matching** launches the **Velocity Picks To CMP Matching** application
- **Export To Half-Link** exports the project database into **Geoshare Half-Link** data interchange format
- **Import From Half-Link** imports the project database from **Geoshare Half-Link** data interchange format
- **Exit** exits the RadExPro program.

Aside from the above, in the bottom right of the window there are three command buttons designed for project structure editing:

- **New** button creates a new area, line, or flow
- **Rename** button renames the existing area, line, or flow
- **Trash** button removes the existing area, line, or flow

The status bar at the bottom of the working window contains hints for mouse usage:

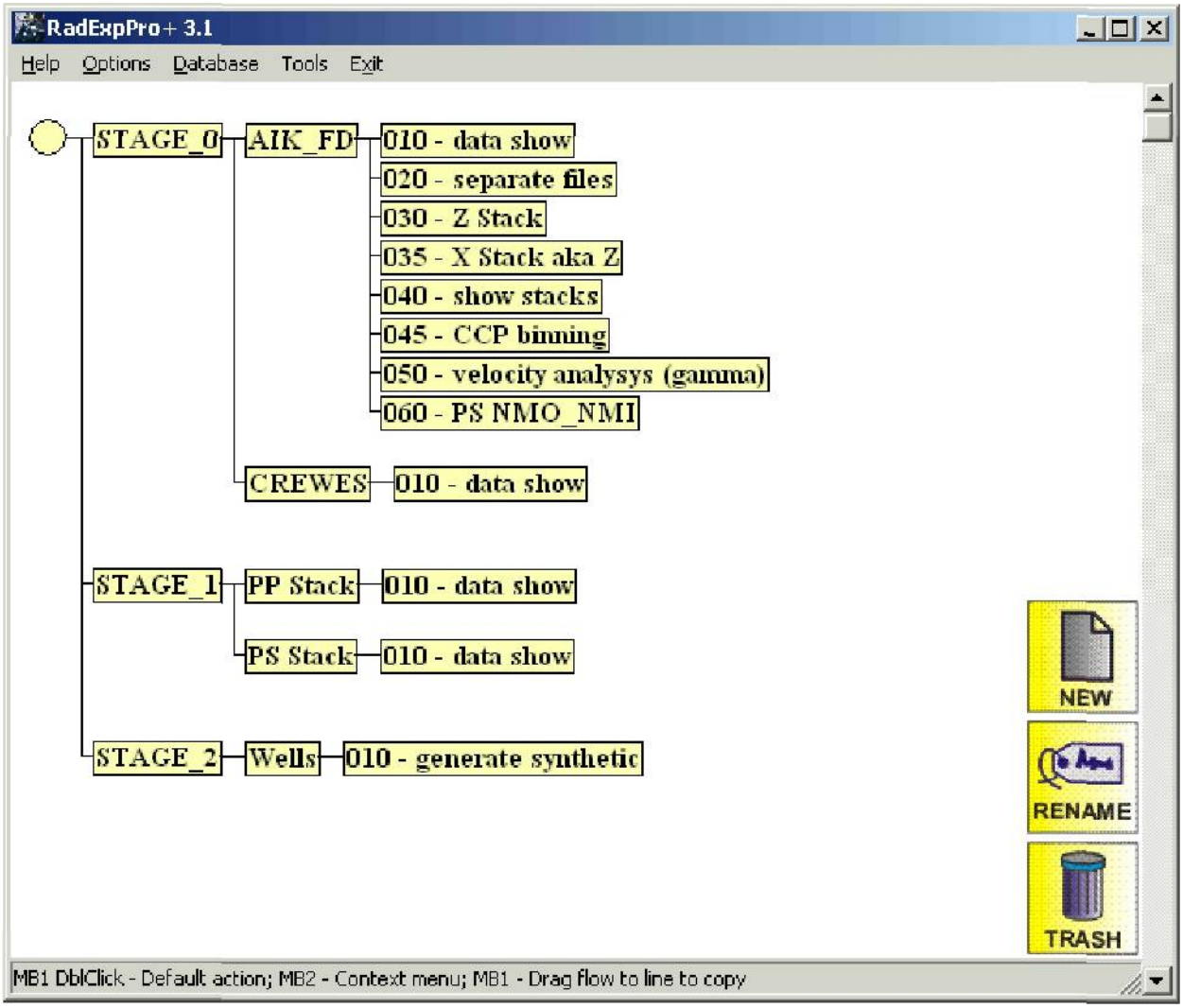
- **MB1 DBLClick** double clicking the left mouse button (**MB1**) on the flow name allows you to edit it;
- **MB2** single clicking of the right mouse button (**MB2**) on the area name, line name, or flow name launches a context menu with available commands;
- **MB1** clicking and holding the left mouse button (**MB1**) on a flow name and dragging it to a line

name allows copying the flow into this line.

Working with project trees

A project tree is a graphical presentation of data organization in the project. All data is divided into areas, lines, and flows. Every area must include one or more lines and every line, in its turn, must consist of flows.

When selecting an existing project, in the main **RadExPro** window. the project tree with areas, projects, and flows will open. This tree can be edited. The names at every level are sorted alphabetically. The window of the already existing project may have the following view:



When creating a new project, a project tree should be constructed. The tree core is a yellow

circle in the upper left corner of the working window.

The tree construction order is the following:

- Create a new area. There are two ways to do this:

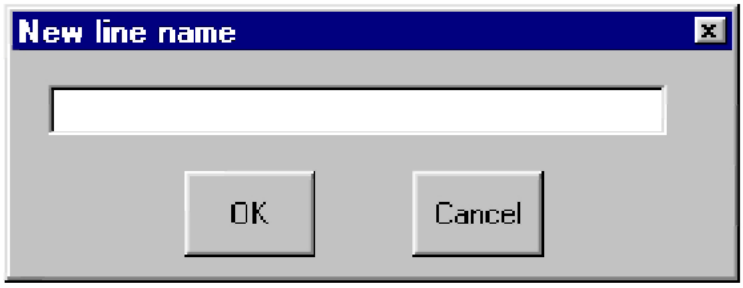
Place the mouse cursor on the yellow circle. Click the right mouse button **(MB2)** and select the **Create new area** command in the context menu. Then, in the opened window enter a name for the new area and click the **OK** button.

Place the mouse cursor on the **New** icon, click the left mouse button **(MB1)** and, holding it down, drag the icon to the yellow circle and then release the button. In the opened window enter a name for the new area and click the **OK** button.

- Create a new line inside the area. There are two ways to do this:

Place the mouse cursor on the area name, click the right mouse button **(MB2)** and select the **Create line** command in the context menu. Then, in the opened window enter a name for the new line and click the **OK** button.

Place the mouse cursor on the **New** icon, click the left mouse button (**MB1**) and, holding it down, drag the icon to the area name and then release the button. In the window that has opened enter the name of a new line and click the **OK** button.



- A new flow is created inside the line. There are two ways to do thCreate a new flow. There are two ways to do this.

- Create a new flow. There are two ways to do this:

Place the mouse cursor on the line name, click the right mouse button **(MB2)** and select the **Create flow** command in the context menu.

Place the mouse cursor on the **New** icon, click the left mouse button (**MB1**) and, holding it down,

move the cursor to the line name and then release the button. In the window that has opened enter the name of a new flow and click the **OK** button.

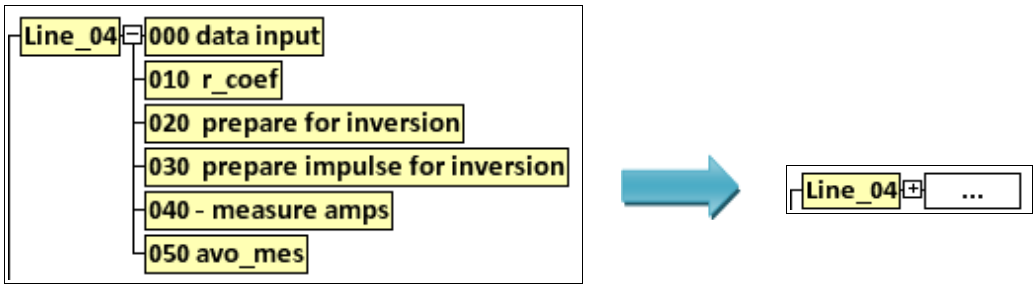
Similarly, areas, lines, and flows can be added to the tree of an already existing project. If necessary, any tree element (area, line, or flow) can be renamed or removed.

In order to rename a tree element, move your cursor over it, click the left mouse button (**MB1**), drag it to the **Rename** icon, and release. Enter a new name for the element in the window that has opened and click the **OK** button.

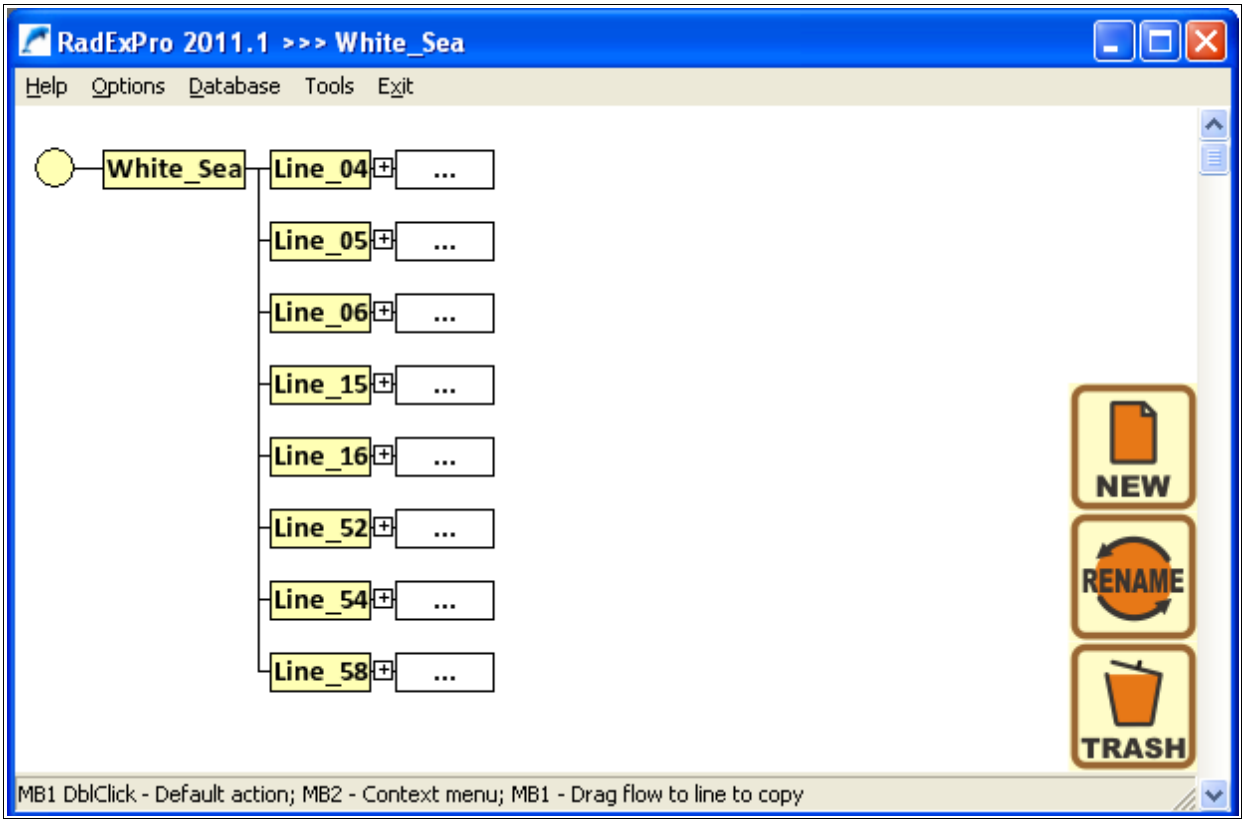
In order to remove the tree element, move your cursor over it, click the left mouse button (**MB1**), drag it to the **Trash** icon, and release. Confirm the removal in the opened window.

ATTENTION: The tree element removal cannot be undone!

When the first flow is created in the profile, a "-" sign appears to the right of the profile name. Clicking this sign collapses all flows in that profile into a single line:

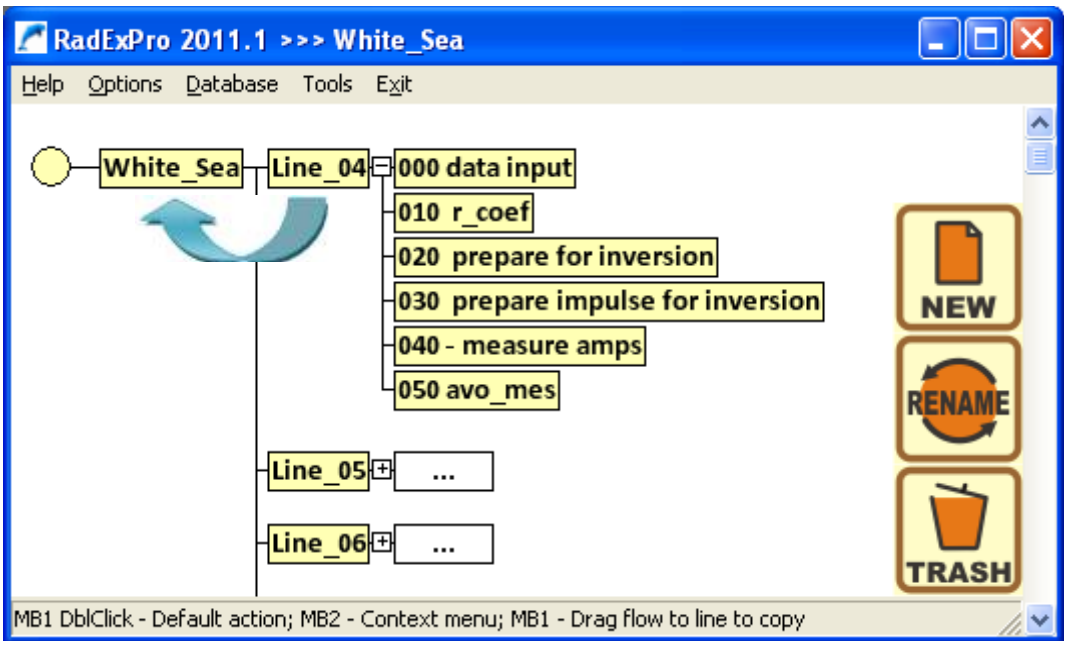


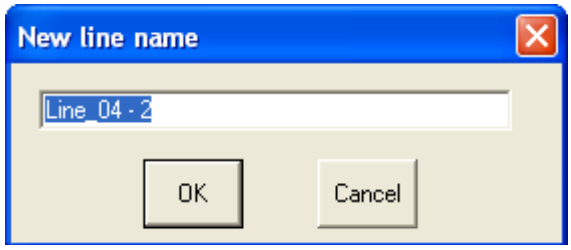
Clicking the “+” sign expands the list of flows. This makes it easier to navigate projects containing a large number of profiles and flows. When all flows in all profiles are collapsed, the project looks as follows:



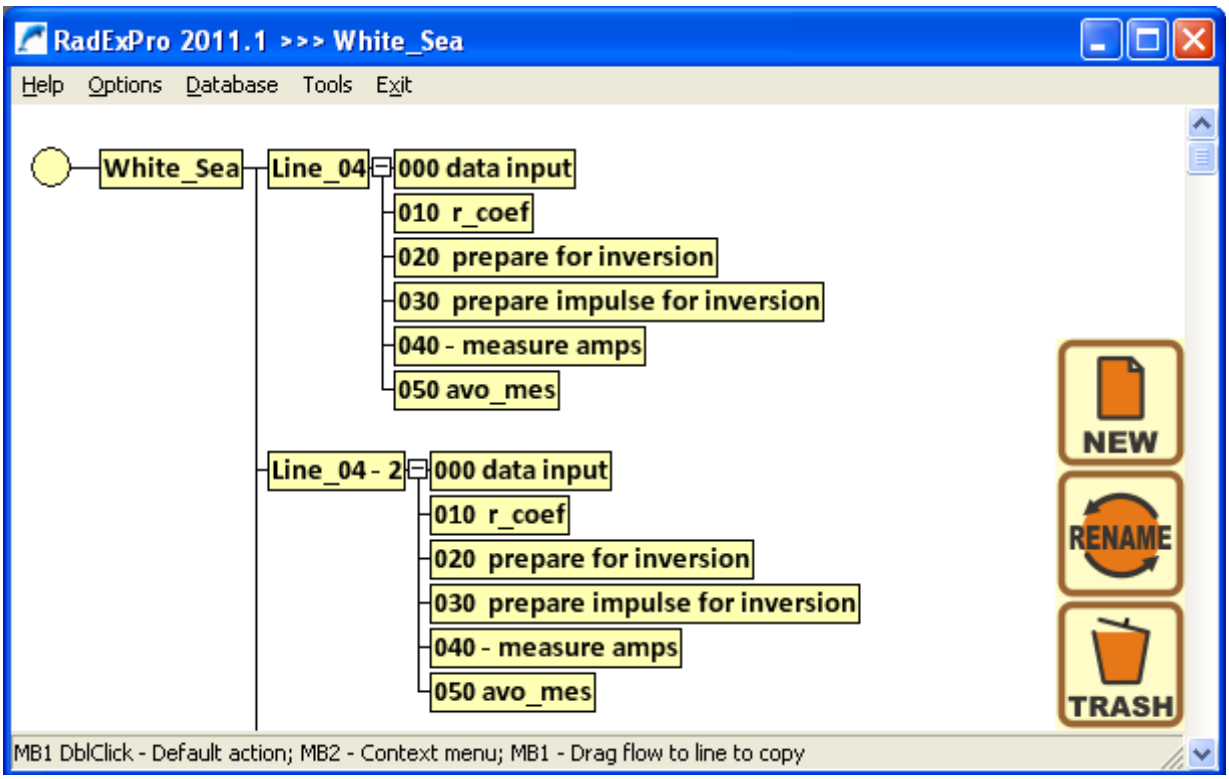
When identical processing procedures are to be repeated for each line, profile copying can make working with the project considerably easier.

To copy a profile, click and drag it with the left mouse button to the area name. The new profile name selection dialog box will appear.





Enter the new name and press **OK**. The profile will be copied with all processing flows and parameters.



Attention: when profiles are copied, all parameters, including all input and output data (data sets, picks etc), remain unchanged. Don't forget to change the names of input and output files and data sets before running the new flows.

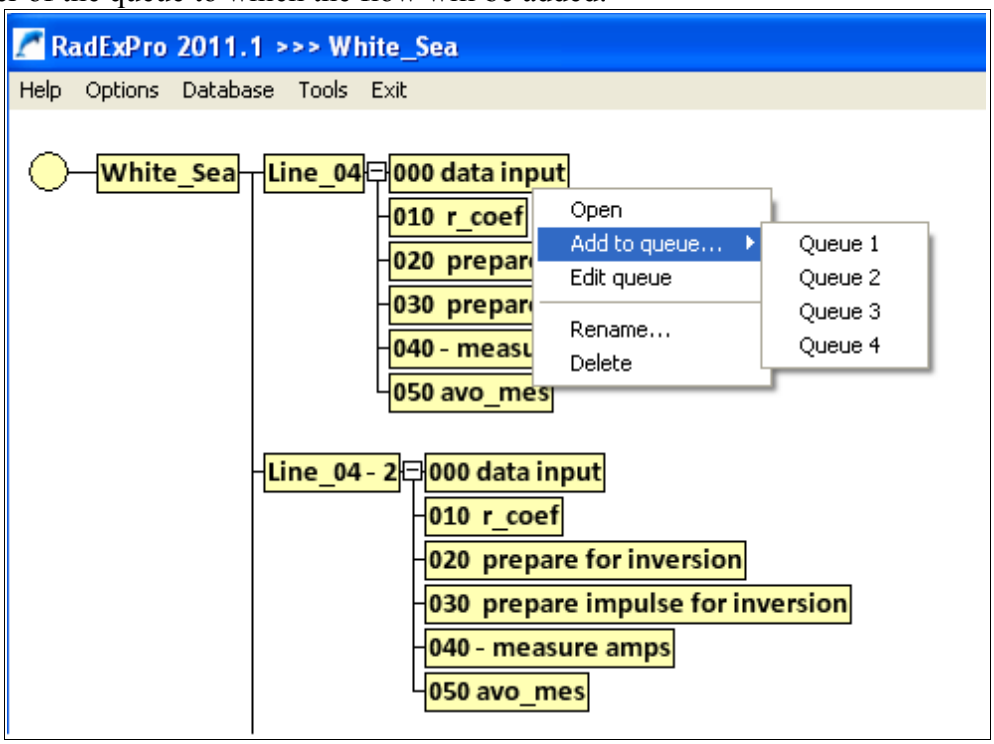
- Project navigation

The project can be scrolled up and down using the vertical scrollbar or the mouse wheel.

- Queues

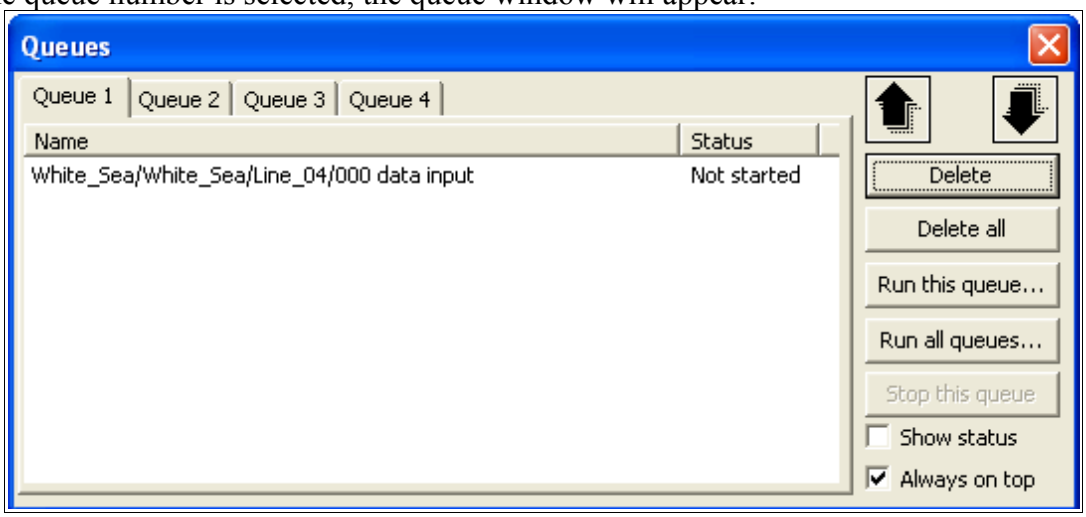
Flow execution can be queued. Both individual flows and entire profiles (i.e. all profile flows one after another) can be added to an execution queue.
To add a flow to a queue, right-click the flow, select the **Add to queue** menu item, and then select

the number of the queue to which the flow will be added:



The program allows running flows in four independent queues. Queues are launched in parallel; this reduces the total processing time on multi-processor or multi-core machines.

After the queue number is selected, the queue window will appear:



The upper part of the window contains tabs that allow switching between the queues.

The current queue editing window shows the list of active flows added to the queue and their status. The flow status indicates the flow processing stage:

Not started – the flow has not yet been started;
Running – the flow is running;
Canceled – execution of the flow has been canceled.
The flows in the queue are executed one after another, from top to bottom. The flow execution order can be changed:

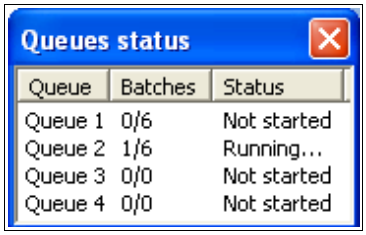
– move the flow up one level in the queue.

– move the flow down one level in the queue.

The right part of the window contains buttons used to manage the list and the flow procedures:

- Delete – delete the selected flow from the queue
- Delete all – delete all flows from the queue
- Run this queue... – run the current queue
- Run all queues... – run all queues
- Stop this queue – stop the execution of the current queue. This button is active only when the queue is running.

Show status – opens a window showing the current status of all queues:



Here:

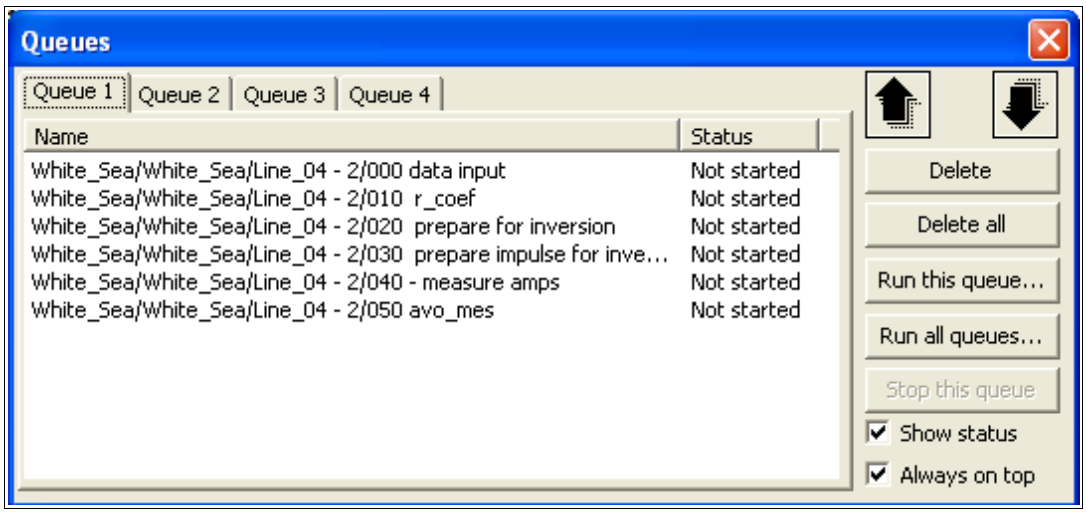
Queue – queue number

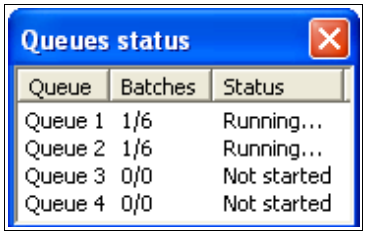
Batches – number of the process in the queue

Status – current process execution status

Always on top – if this option is enabled, the queue window will always be displayed on top of other program windows.

An example of queues is shown below. Two queues were created; each of them contains a set of flows in separate profiles. When all queues are launched, the two current queues will be run in parallel.


Queues
Queue 1
Queue 2
Queue 3
Queue 4
Name
Status
White_Sea/White_Sea/Line_04 - 2/000 data input
Not started
White_Sea/White_Sea/Line_04 - 2/010 r_coef
Not started
White_Sea/White_Sea/Line_04 - 2/020 prepare for inversion
Not started
White_Sea/White_Sea/Line_04 - 2/030 prepare impulse for inve...
Not started
White_Sea/White_Sea/Line_04 - 2/040 - measure amps
Not started
White_Sea/White_Sea/Line_04 - 2/050 avo_mes
Not started
Delete
Delete all
Run this queue...
Run all queues...
Stop this queue
Show status
Always on top


Queues status
Queue
Batches
Status
Queue 1 1/6 Running...
Queue 2 1/6 Running...
Queue 3 0/0 Not started
Queue 4 0/0 Not started

Working with flows

Data processing and visualization is performed by creation of execution sequences (flows) from the set of available modules. These flows are stored inside the database and can be changed, copied, or removed.

Such organization has a number of advantages because it allows storage of the processing history, which in turn allows you to return to any step and repeat the whole process, with different parameters or reuse the already created processing flow with another data set.

The processing sequence (flow) is a set of processing modules, whose inputs and outputs are somehow connected.

RadExPro has two modes of flow presentation: simple (text) mode and complex (graphical) mode.

In the simple (text) mode the processing flow is represented as a list of processing modules so that the output of every module is connected to the input of the module that follows it. The flow is conducted sequentially (top-down) and linearly.

In the complex (graphical) mode the processing modules are presented as icons and the links between them as tubes. The inputs and outputs of the processing modules are manually connected by tubes. The modules can have several inputs and outputs. The processing flow can branch out and form cycles. Such presentation of the flow gives more flexibility in data processing organization. Owing to this, the flows created in graphical mode cannot always be presented in graphical mode.

Every flow starts with one of the modules of data input from the **Data I/O** chapter. Usually (for files registered in the project) it is **Trace Input.** A data input module initializes the flow and allows selection of data for processing. There could be several data input modules within one flow. This can be useful, for example, for:

- combining several data sets for co-processing;
- data set processing by different methods with succeeding co-visualization by **Screen Display**

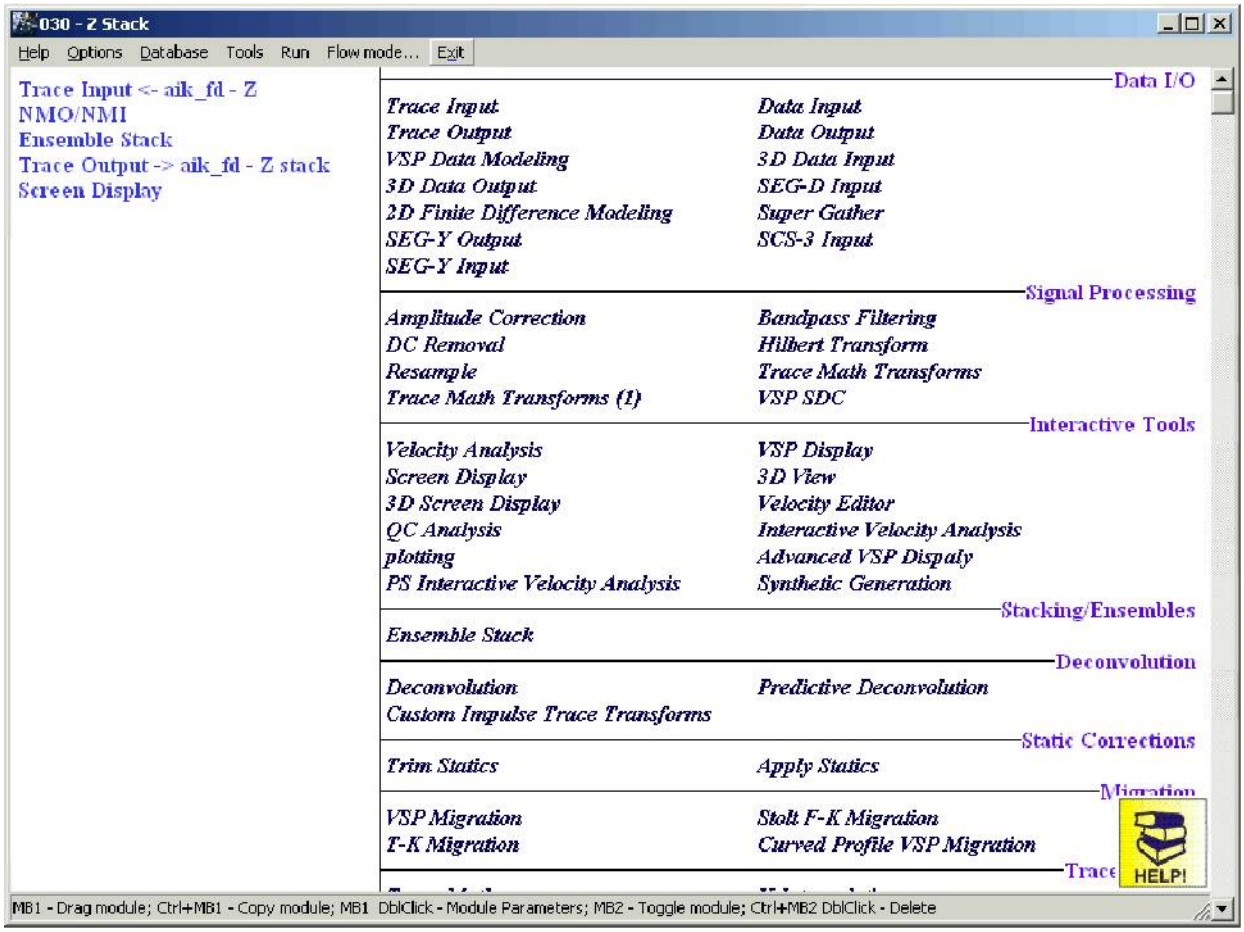
module or combining in one output file.

There are also modules designed for data editing and processing: amplitude correction, filtering, stacking, etc. Every flow usually ends with a data output module, saving processing results on hard-disk (for instance, **Data/Trace Output)** and/or by interactive tools (for instance, **Screen Display).**

The result of every processing routine/module can be adjusted by changing the parameters of the module.

Flow construction (simple mode)

When you double-click the left mouse button (**MB1**) on the flow name in the project tree, the flow working window will open. If simple (text) representation of the flow is possible, then by default the flow will open in simple mode. The window is divided into two parts: the right one is a list of available routines (modules) which can be used to create a flow; in the left part the flow itself, i.e. the modules used in the active flow, are displayed.



The flow allows you to conduct various manipulations with the seismic data registered in the project (for example, processing, correction, interactive analysis). Manipulation of external files is also possible.

The modules from the right part are grouped by chapters. For example, **Data I/O** (data input/output), **Signal Processing, Interactive Tools,** etc.

You can include a module into the active flow by two methods:

- Click the left mouse button **(MB1)** on the name of the desired module in the list of available routines in the right part of the window and, holding it down, drag the mouse cursor to the left part of the window and release the button (drag-and-drop).

- Double-click the left mouse button **(MB1)** on the left side of the working window and select the required module from the list of available routines that will open as a pop-up menu;

When a module is being added into the flow, a dialog box with the module parameters will appear. The way it looks depends on the module selected.

When creating a flow the following operations are possible:

- **Switching module state in the flow (active/inactive)** - is performed by clicking the right mouse button (**MB2**) on the name of the module in the flow. Inactive modules will be displayed in narrow italics, and three * (***) will appear before its name

- **Transfer of a module in the flow** - is done by clicking the left mouse button **(MB1)** on the module name and, holding it down, moving the cursor to a new position and releasing the button

- **Viewing and editing of module parameters** - is performed in a special dialog window that opens when you double-click the left mouse button **(MB1)** on the module name in the flow

- **Module duplication** - is done by means of the keyboard combination Ctrl+MB 1 on the module

name

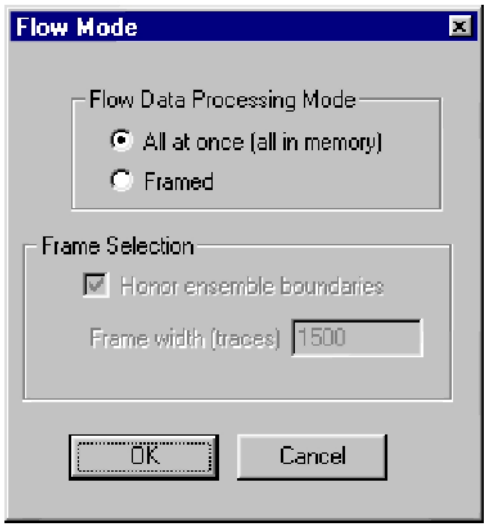
- **Module removal** - is done by double-clicking the right mouse button (**MB2**) on the module name while simultaneously holding the Ctrl. key down. When doing this, the module with set parameters will be removed from the flow and placed onto the clipboard

- **Restoring the last removed module from the clipboard** - is done by means of the keyboard combination **Shift+MB1** in the flow construction window. The removed module can be restored not only in its initial flow but also in any other flow of this project

In the working window menu the **Help, Options, Database, Tools** elements are the same as in the main window of the program (see **How to Start).** The **Exit** command of the menu, unlike the similar command in the main window, does not result in the exiting of the program. The selection of this command allows closing the flow window and returns the user to the project tree viewing, i.e. to the main **RadExPro** window. The menu also contains the following commands specific to the flow window:

- **Run** launches active flow execution

- **Flow mode** allows the user to specify the mode of memory usage during flow execution.



When selecting this command the following window will be displayed:

The **All at once (all in memory)** mode is set by default. When using this mode all data from the flow is placed into system memory. This mode is used for small data files. For large data files the **Framed** mode is used. When using this mode all data from the flow is processed in portions, their volume in traces is defined in the **Frame width (traces)** window. Activation of the **Honor Ensemble boundaries** option avoids trace division inside the ensemble.

Complex (graphical) mode of flow creation

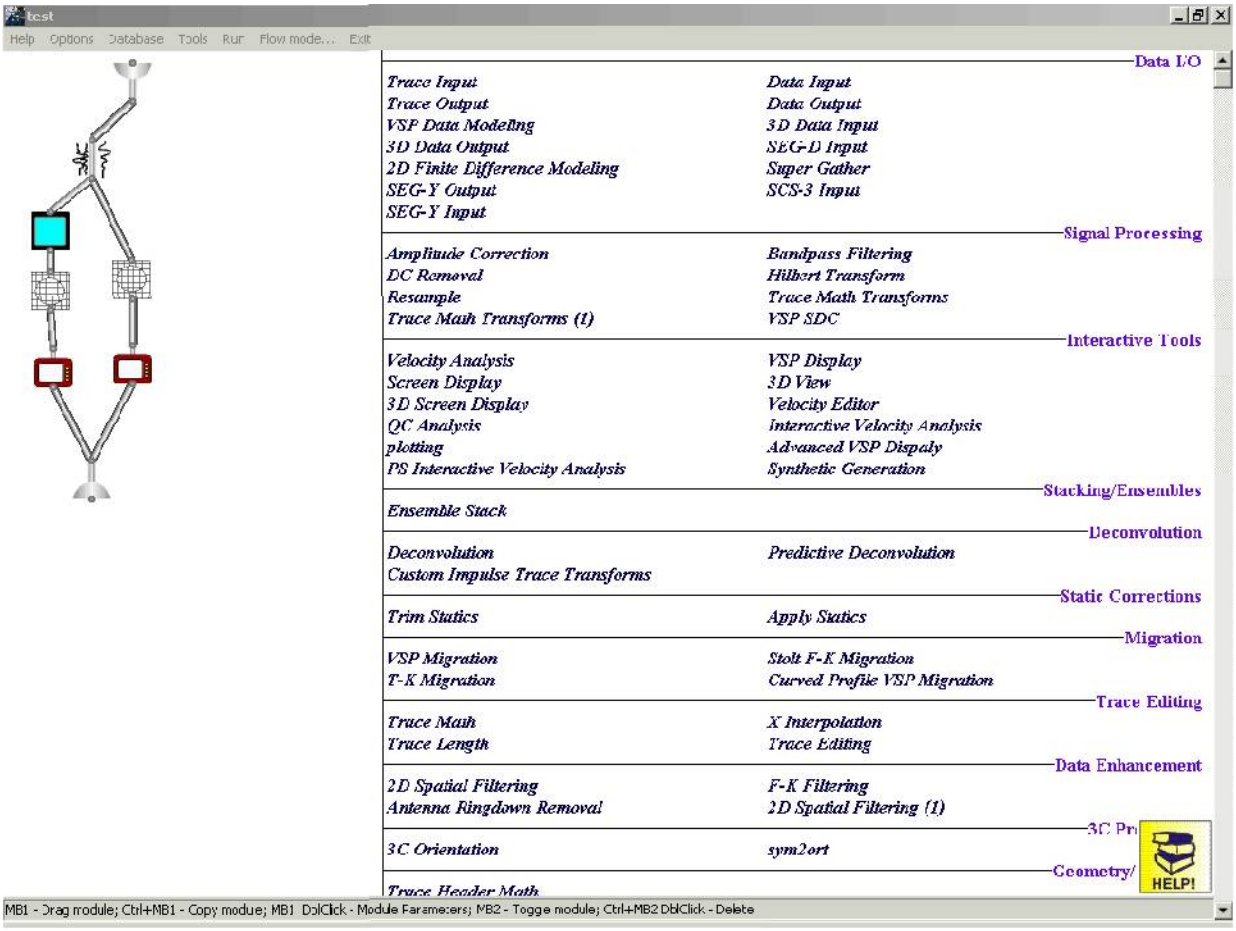
In the graphical mode the left side of the working window contains a graphical presentation of the flow, and the right side contains the set of available processing modules.

Every module included in the flow is presented as an icon.

Module inputs and outputs are connected manually. The module input is displayed as a small circle located in the upper part of the icon; the module output is displayed as a small circle located in the bottom part of the icon. When the mouse cursor is placed on the icon a small hint with the module name will appear on the screen.

In the case when several tubes are connected to the module input, all data from these tubes will be combined (in order of tube creation). In the case when several tubes are connected to module output, all data "out-flowing" from the module will be distributed to every tube.

Creation of the flow in graphical mode comprises modules adding to the flow and connecting them by tubes. Later the modules can be added to the flow, temporarily excluded from it, copied onto the clipboard and/or pasted from it.



- **To add a module into the flow,** drag it from the list or double-click on the left side of the flow window and select the desired module from the pop-up menu.

- **To connect the input of one module to the output of another,** click the right mouse button (**MB2**) at the input (or output) of one module and drag the tube that appears to the output (or input) of another module. When dragging, the line showing the tube will follow the mouse cursor. When the cursor points to a possible connection site, the tube-line will be thicker. At this moment, you can establish the connection by releasing the right mouse button. It is impossible to connect input with another input or output with another output.

- **To change the parameters of the module already included in the flow,** you can simply double-click the left mouse button (**MB1**) on the module's icon in the flow, where upon the dialog box for parameter specification will appear.

- **To move the module icon,** drag it by holding down the left mouse **(MB1) button**.

- It is possible to exclude the module from the module sequence under execution but keep it in the flow (make the module inactive). Inactive modules are slightly faded. To enable **switching the module mode in the flow (active/inactive),** click the right mouse button (**MB2**) on the module's icon.

- **To remove the module from the flow,** double-click the right mouse button (**MB2**) on the icon of the module in the flow while simultaneously pressing the Ctrl. key. The module with set parameters will be be removed from the flow and placed onto the clipboard.

- **To remove all tubes connected to the module input or output,** click the left mouse button **(MB1)** on the corresponding small circle in the upper or bottom part of the icon of the module in the flow.

- **To copy the module onto the clipboard,** click the left mouse button **(MB1)** on the center of the icon while simultaneously pressing the Ctrl. key.

- **To paste the module from the clipboard,** click the left mouse button **(MB1)** on the left side of the screen while simultaneously holding down the Shift key.

In the graphical mode the menu of the flow working window contains the same commands as in the text mode (see **Flow construction (simple mode)** chapter).

Switching between flow editing modes

When creating a new processing flow (or when opening an already existing flow, if it can be viewed in text mode) a simple presentation mode is selected by default. To switch to the complex mode, double-click the right mouse button (**MB2**) on the left side of the window. The flow will be converted and presented graphically but the routine's sequence will be the same as in the flow presented in simple (text) mode.

Simple (text) flow is executed sequentially and linearly. Graphical (complex) mode can have branches and cycles. In this case a simple (text) presentation of such a flow is impossible and when opening this flow it will be displayed graphically.

If the flow is displayed in graphical mode but it does not have branches and cycles, then it also can be displayed in a simple (text) mode. To switch this kind of flow into simple mode, double-click the right mouse button (**MB2**) on the left side of the window.

Flow execution

The Run command of the working window serves to execute the flow. A small status bar will appear on the screen. On the status bar the list of active modules of the flow and task completion rate for every module will be shown.

At present, in the flow there are several interactive modules meant to interact with the user during their execution (for example, **Screen Display, VSP Display,** etc.). Interactive modules exist in a separate window and can be considered by the user as independently executed tasks. Via standard Windows functions, it is possible to switch from the interactive module back to the **RadExPro** working window and continue working with it. At the same time the flow containing the interactive module is still under execution and the interactive module window is available for visualization and activation. Thus, more than one flow can be executed. This is a very convenient feature because it allows you to compare the data on the screen. The interactive modules are completed either through the module interface or at the completion stage of the whole program (upon closing of the main working window). In either case the flow is executed to the end. When launching more than one module at once, remember that every launched flow needs computer memory (mainly for data storing) and that is why a limited number of flows can be launched at the same time.

It is possible to interrupt flow execution. To do this, close the status bar and click the **OK** button when the corresponding question appears. You should use this opportunity only as a last resort because such interruption is abnormal and may result in loss of system resources (of course they will be restored after a Windows restart).

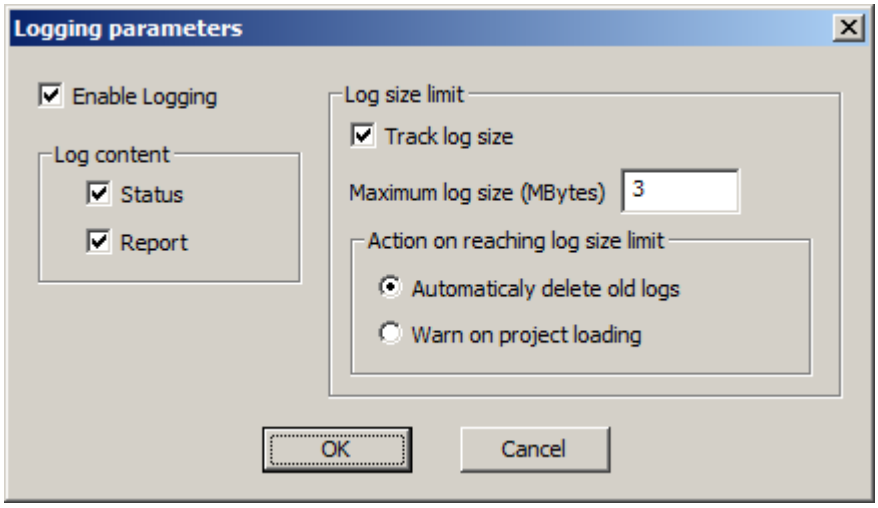
Logging

The **RadExPro** program includes the facility of logging of flow execution. When a flow is being executed, appearing messages can be written to a text file (log). When necessary, this file can be viewed by any text editor. It could be helpful to look into the log, when the flow execution terminated abnormally. The log could explain the reason of the error, or at least define at what stage

the problem occurred. The logging facility is optional, i.e. it can be switched off. Logging parameters can be set through a dedicated dialog window. All logs are stored in the LOGS subfolder of the root folder of the project. The structure of the LOGS folder repeats that of the project, so the logs corresponding to a certain flow are stored in the folder of the same name. Each flow execution creates a new log. The name of the log consists of the date and time of launching of the flow execution, to distinguish the logs related to one flow that was executed several times. The name structure of a log is **yymmddhhmmss.txt** (For example, the name of the log **090101001053.txt** indicates that the flow was executed on the 1st of January, 2009 at 00 hours 10 minutes 53 seconds.)

The **Logging parameters** dialogue can be opened using the **Options->Logs...** entry of the main program menu.

The logging parameters:



- **Enable logging –** enables/disables logging facility.

- **Log content** – defines message types to be included into the log:

Status – doubling of the messages from the **Status Window**

Report – dedicated messages, created specially for logging.

- **Log size limit -** this group combines parameters controlling the size of the LOGS folder:

Track log size – enables/disables log folder size tracing.

Maximum log size – sets maximum log folder size in MBytes

Action on reaching log size limit –defines the action to be performed, when log folder size reaches maximum. Select one of the following:

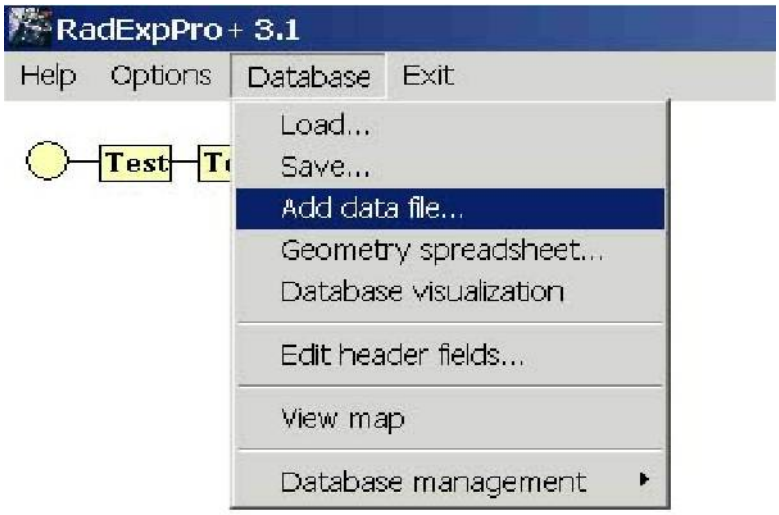
Automatically delete old logs

Warn on project loading Adding data files to the project

Though **RadExPro** provides for the possibility of working with external (in respect to the project) data files, in most cases any operations with a data file should be started with its registration in a

database of the project so that you can use the program's features in full measure.

There are two ways of adding a data file to the project. Having created a **RadExPro** project, area, line, and flow, it is possible to have external (in respect to the project) files registered by means of entering it into a flow via one of the data input routines (for example, *SEG-Y Input)* and then saving it in the project via the *Trace Output* routine. However, in most cases the alternate method is more convenient. Select the Database menu item in the main RadExPro window (project tree window) or in the working window of any flow within the project and then select **Add data file...** in the submenu that has appeared.

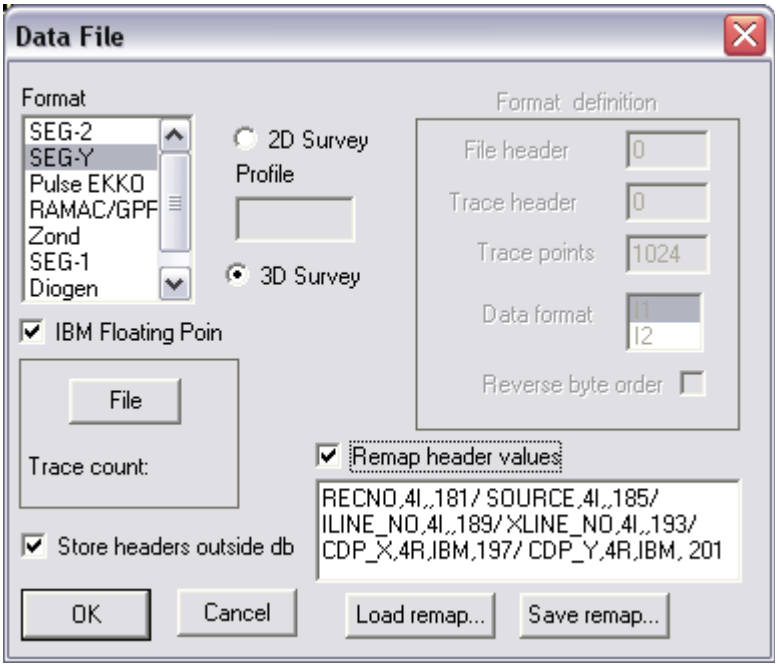


This command activates a special tool for adding data files.

Adding data files to the project

Adding a data file

When the **Add data file...** command of the Database menu is selected, the **Data File:** dialog box will appear.



In the Format field, select the format of the file to be registered in the project.

Supported data formats

At present the system is compatible with 7 standard data formats (SEG-Y, SEG-2, SEG-1, Pulse EKKO, RAMAC/GPR, Zond, Diogen) as well as with **User Defined** format**.** The user defined format support allows reading the data presented in any demultiplexed format with fixed length of file header and trace header, as well as fixed number presentation and byte order throughout the data.

When selecting one of the standard formats, **RadExPro** fills in the project database header fields automatically on the basis of standard data location in the headers of the format selected. If the format of the source file is not completely compatible with the accepted standard you can apply format remapping **(Remap header values).** When selecting a user defined format always apply remapping to correctly fill in the header fields.

Dialog parameters

- **Format** - select a file format form the list of formats

- **2D Survey, 3D Survey** - the switcher that defines file identification in project as 2D or 3D surveys

- **Profile** - in this field you can define the sequence number of 2D profile if 2D **Survey** is selected

- **IBM Floating Point** - if this option is activated the amplitudes (trace counts values) will be read from the file as 32-bit real numbers with a floating point in the IBM Floating point format.

- **Store headers outside db** - if the option is off, the headers of the file under registration will be stored directly in the project database file. Otherwise a separate file will be created inside the directory for storing the headers. If there is much data in the project, the storage of headers in separate files increases program operation speed. This option is activated by default.

- **Format definition** - a group of parameters of the user defined format. These parameters are available only when selecting the **User Defined** format from the **Format** list:

- **File passport** - in this field, define a file header size expressed in bytes (0 - if the file header is lacking)

- **Trace passport** - in this field, define a file header size (0 - if the file header is lacking)

- **Data format** - in this field, select the format of trace samples in the input file. Possible variants:

I1- 8-byte integer

I2- 16-byte integer

I4 - 32-byte integer

R4 - 32-byte real with floating point

UI1 - unsigned 8-byte integer

UI2 - unsigned 16-byte integer

UI4 - unsigned 32-byte integer

If R4 format is selected in the field, the data will be read either as an **IBM floating point** or as IEEE, depending on the **IBM Floating Point** switch status. For any other **Data format** header the **IMB Floating Point** status is disregarded.

- **Reverse Byte order** - if this option is activated, the program will assume that all values in the file are recorded with reverse byte order

- **Remap header values** - if this option is off, the header fields in the project database will be filled in automatically on the basis of the standard definition of the selected format. If the option is on, the header fields will be filled in accordance with format remapping. If the option is active then a text field for remap specification as well as **Save remap...** and **Load remap...** buttons allowing you to save the active remap in a database and to load the already saved remaps, respectively, via the **Database Object Selection** (see **Database Object Selection window** chapter) become available.

Example of format remapping:

RECNO,4I,,181/ILINE_NO,2I,,189/CDP_X,4R,IBM,197/CDP_Y,4R,IBM,201

In this example:

- RECNO = database header field name where the values from the trace header field specified in a remap will be recorded;

- 4I = a **field format** in a trace header, i.e. a format in which the value is stored in the source file, in this case it is a 32-byte integer.

- IBM or IEEE — flag, identifying the representation standard used for the real floating point-numbers. Used for the 4R format only. For the integer types the flag is not used, and nothing is placed between the two commas.

- 181 = offset of the required field in the trace header expressed in bytes;

A **field format** may contain the following maps:

1I - 8-byte integer;

2I - 16-byte integer;

4I - 32-byte integer;

4R - 32-byte floating-point real (either IBM or IEEE standard depending of the flag that follows);

8R – 64-byte floating-point real.

Thus, **RECNO,4I,,181** means that RECNO header field corresponding to the receiver point number will be filled by 32-byte integer values read from trace headers starting with the 181th byte from the beginning of the trace header.

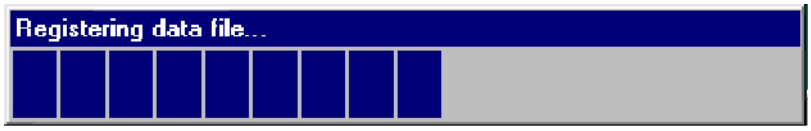
File - After format and other parameters have been selected, click a **File** button and select a data file which should be registered in the project from the file selection window. The number of traces contained in the file will appear in the **Trace count** field directly under the button. The trace number will be computed on the basis of the format and remap selected. If a file is not compatible with the format selected, then a trace number may be populated incorrectly.

After all parameters and the file name have been defined, click the **OK** button. A **Database Object Selection** (see **Database Object Selection window** chapter) window will open where the user will be asked to enter a name under which the data set from the source file **(dataset)** will be registered in the database and to select a database level where the data set from the source file will be stored.

We recommend that all seismic data be stored in the database at the **Line** level. Select one of the

database sections of **Line** level and specify the name for a new data object under which the file will be registered in the project, and click the **OK** button. Wait until the selected file is registered in the database.

The process of registration will display as a progress bar.

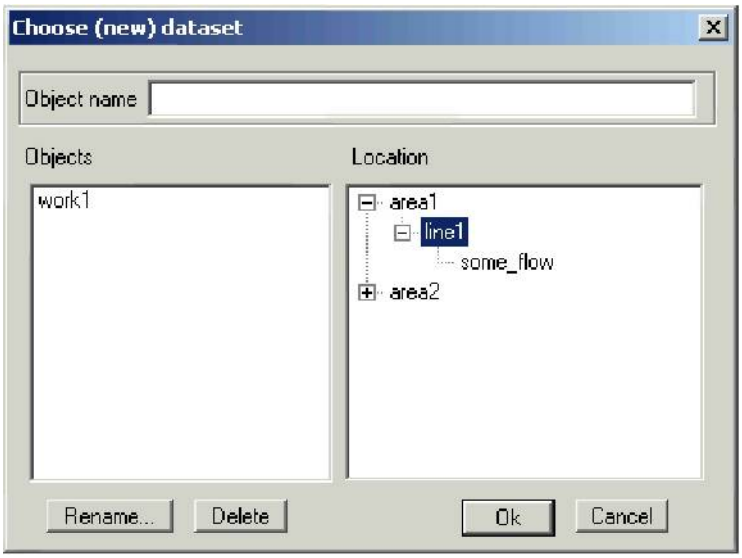


Database Object Selection window - saving and opening project database objects

The Database Object Selection window in the **RadExPro** program is a standard tool for operating with database objects similar to open/save file dialog box which is a standard tool for manipulation of files in a Windows OS.

Almost every time you choose any existing **RadExPro** project database object or when saving a new object to database you will see the **Database Object Selection** window. The header of the **Database Object Selection** window **will** contain an enquiry which will depend on the task under execution.

Below you can see the **Database Object Selection** window that appears while a new data file is being registered in the project:

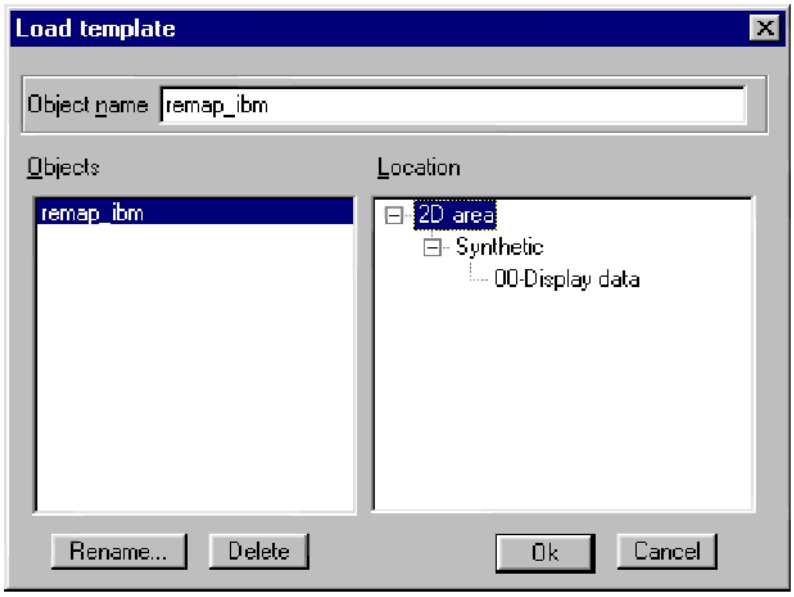


The project tree is shown to the right in the **Location** field, and the active section is selected. In this window the user can navigate throughout the database structure choosing the desired section. All objects of this kind registered in the active DB are displayed In the **Objects** field to the left. At the top is the **Object name** field, in which the name of the selected object is displayed.

In this example the project database contains 2 sections of Area level *(area1* and *area2),* in the *area1* section is one section for **Line** level *(line1),* which, in its turn, contains one section for **Flow** level, i.e. a flow *(some_flow).* In the current section there is one data file, *work1* that has already

been registered. In the Object name field the user is asked to enter a name under which the new file will be registered in the project.After the name has been selected and the **OK** button is pressed, the data will be registered in the current project database section (in this example that would be *line1)*. In order to register a file in another database section, change the current section.

Similarly, when opening any existing database object, you should first select the DB section, containing the required object, and then select the object from the list of already existing objects of this kind in the section. Below, is an example of the **Database Object Selection** window appearing while loading the previously saved format remap:



In this example, in the active section of Area level *(2D area)* there is only one remap of *remap ibm* format and this is the remap chosen by the user for loading.

The **Database Object Selection** window is used in different RadExPro program modules for opening/saving of different types of data objects. Therefore, in the **Objects** field, only the objects to which the current operation can be applied are shown. Thus, when selecting a file with a seismic dataset registered in the project, only the datasets that belong to the active level will be shown in the **Objects** field. Objects of another type (for example, picks) belonging to the same database level will not be displayed.

You will also see two buttons in the **Database Object Selection** window:

Rename - is used to change the name of the selected object. To do this, select the desired object in the **Objects** list, enter a new name in the **Object name** field, and click the **Rename** button. After the

operation has been confirmed by the user, the name of the selected object will be replaced by the new one.**Delete** - is used to delete the object from the database. To do this, select the desired object from the **Objects** list and click the **Delete** button. After the operation has been confirmed by the user the object will be deleted from the project database.

ATTENTION: if the selected object is a set of seismic data, only the file registration in the project will be canceled, but the file will remain on disk. Objects of any other type are stored directly in the project database and if they are deleted from the database they are also deleted physically. This operation cannot be canceled or undone!

Geometry Spreadsheet - spreadsheet editor for trace header values

The Geometry Spreadsheet application is meant for spreadsheet visualization of header field values of a data set in the project and for performing operations with them.

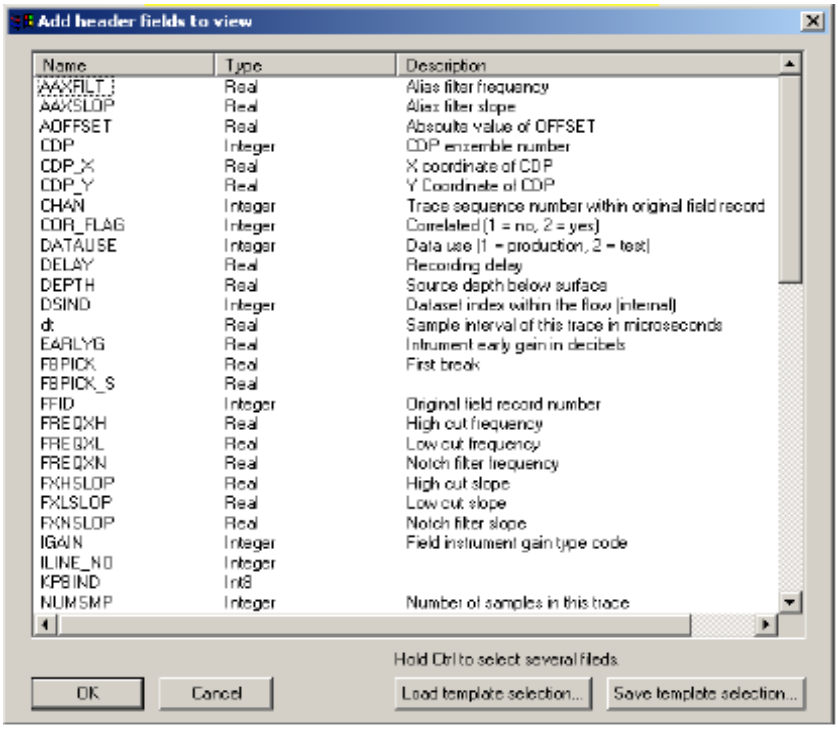
To launch the application, select a **Database/Geometry spreadsheet** command either from the main program window (project tree window) or from the working window of a flow. After a data set has been chosen in the **Database Object Selection** dialog box that appears, the **Geometry spreadsheet** application window will open. The window will contain one column with TRCENO header field values (trace numbers) for a selected data set.

Using the geometry spreadsheet editor

The Geometry spreadsheet editor menu includes the following options:

- **Fields -** opens the commands list, that allows you to add and remove columns from the table.

Add field – adds one or several columns to the table with the values of chosen fields. The choice of header fields is performed in the pop-up window.



In order to choose several header fields, select them with the mouse, holding the Shift or Ctrl button. The buttons **Save template selection** and **Load Template selection** allow you to save the chosen headers set as a template in the database and load the templates that have been saved earlier.

- **Hide field –** hides the selected column from the table.

- **Edit –** opens the list of the following commands.

- **Save changes –** saves all changes to the fields carried out during the work of the application.

- **Undo all changes –** cancels all the changes.

- **Tools –** opens the commands list, that allows different operations with headers, import/export of the data.

- **Header Math –** performs mathematical operations with the header values (see the paragraph "Using the mathematic operations headers editor")

- **Export...—**exports the content of the spreadsheet to a file in ASCII format

- **Import... –** imports the file in ASCII format. You should specify the text rows and columns to import into the spreadsheet (see the paragraph "Import of files in ASCII format").

- **Import SPS X...—**allows import of the data to the header fields from the SPS-X files (see the paragraph "Import of SPS-X files").

- **Import UKOOA p1-90...** allows import of UKOOA p1-90 files

- **Trace Enumerator –** invokes the tool for the specification of continuous trace numbering, sorted by two header fields (see the paragraph "Trace Enumerator tool")

- **Exit –** leads to exit from the geometry table. If there are some changes to the header fields during the work, there will be a saving request.

The following buttons and keys are used during the editing of the spreadsheet:

- Single left mouse button click **(MB1)** on a header column selects the whole column. In order to select several columns, select them one by one, holding the Ctrl button **(Ctrl+MB1)**.

- Double left mouse button click **(MB1)** on a header column sorts the values in the column in ascending order

- Double left mouse button click **(MB1)** on a cell allows text editing

- Single left mouse button click **(MB1)** on a cell selects the cell; use the left mouse button holding the **Shift (Shift+MB1)** in order to select the values set in the column or several columns.

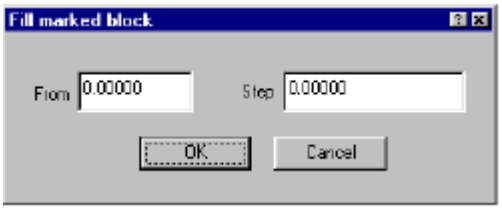
- The selected values set can be copied to the clipboard using the **Ctrl+C** buttons.

- Select the cells and use the buttons **Ctrl+V** to insert the values from the clipboard into the cells. If the cells' size in the clipboard exceeds the selected block that you are going to paste, only a part of the values from the clipboard will be inserted.

- **Ctrl-PageUp**: jumps to the start of a column

- **Ctrl-PageDown**: jumps to the end of a column

- **Ins**: fills in the selected block with new values. You should enter the start value



in the field **From**, while the increment is entered in the field **Step**.

- **F**: finds the given value in the current column.

Using the editor of mathematic operations with headers

The module **Trace Header Math**, designed for mathematical operations with header values, is activated using the command of the geometry table editor window **Tools/Header Math…**

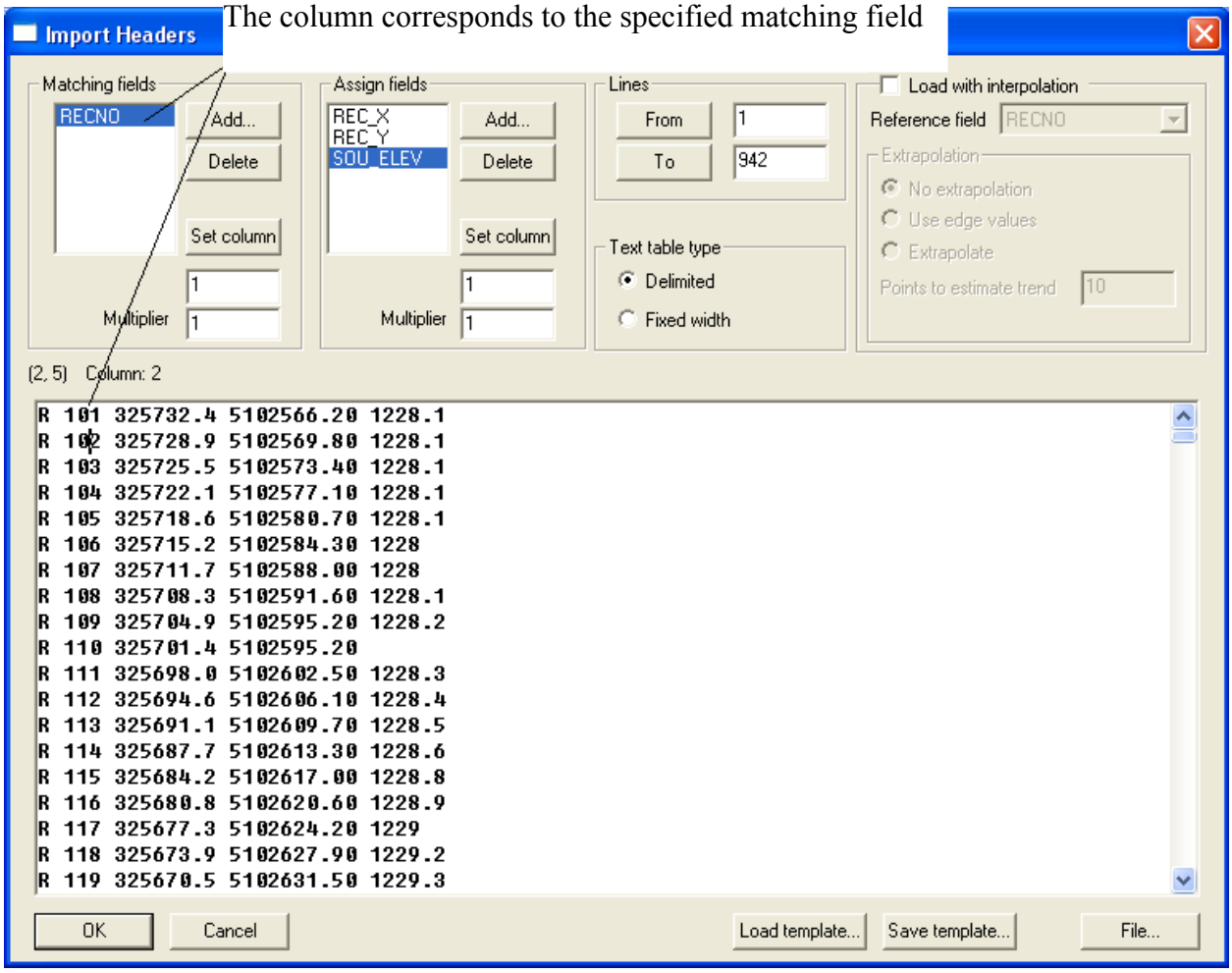
The module description is given in the paragraph **“Processing modules” → “Geometry/Headers” → “Trace Header Math**” of the present manual. The single distinction is that in this module the mathematical operations are applied to the headers in the table immediately after you click the **OK** button in the module's parameters window.

Import of ASCII files

When the **Tools/Import** command is selected from the menu of the **Geometry Spreadsheet**

window the **Import headers** dialog box appears:

Import parameters



- **File...** - opens a standard window for ASCII spreadsheet file selection from which the values will be imported

- **Matching Fields** - shows the header field for which existing values will be compared to the values of the column selected for this field, while importing the columns selected in the **Assign Fields** field into corresponding header fields

- **Assign Fields** - shows the name of the headers to which the values imported from the columns will be assigned.

The **Matching Fields** and **Assign Fields** correlation is shown in the following example: if you set the parameters the way it is shown in the Figure above, then in all traces with a field value RECNO equal to 40 the REC X field values will be replaced by 413291.0. In traces with a field value RECNO equal to 41 the REC_X field values will be replaced by 413289.0, etc., i.e. the

settings in the **Matching Fields** field set a fitting criterion. When fulfilling this fitting criterion the import is accomplished according to the **Assign Fields** field settings.

Both the **Matching Fields** and **Assign Fields fields have** the following buttons:

- **Add...** - this button adds a header name from the headers list in the field

- **Delete** - this button deletes a header name from the field

Individually for every header in the lists of **Matching Fields** and **Assign Fields** fields the following parameters can be set

- **Column** - in this field a column number is shown in a text file which, at this moment, is correlated with a selected header. To change the conformity of headers and columns: (1) select a desired header, (2) click the left mouse button (MB1) on the column that should be correlated with this header and (3) click the **Column** button. In the **Column** field a new column number will appear. Also the value in the field can be changed manually

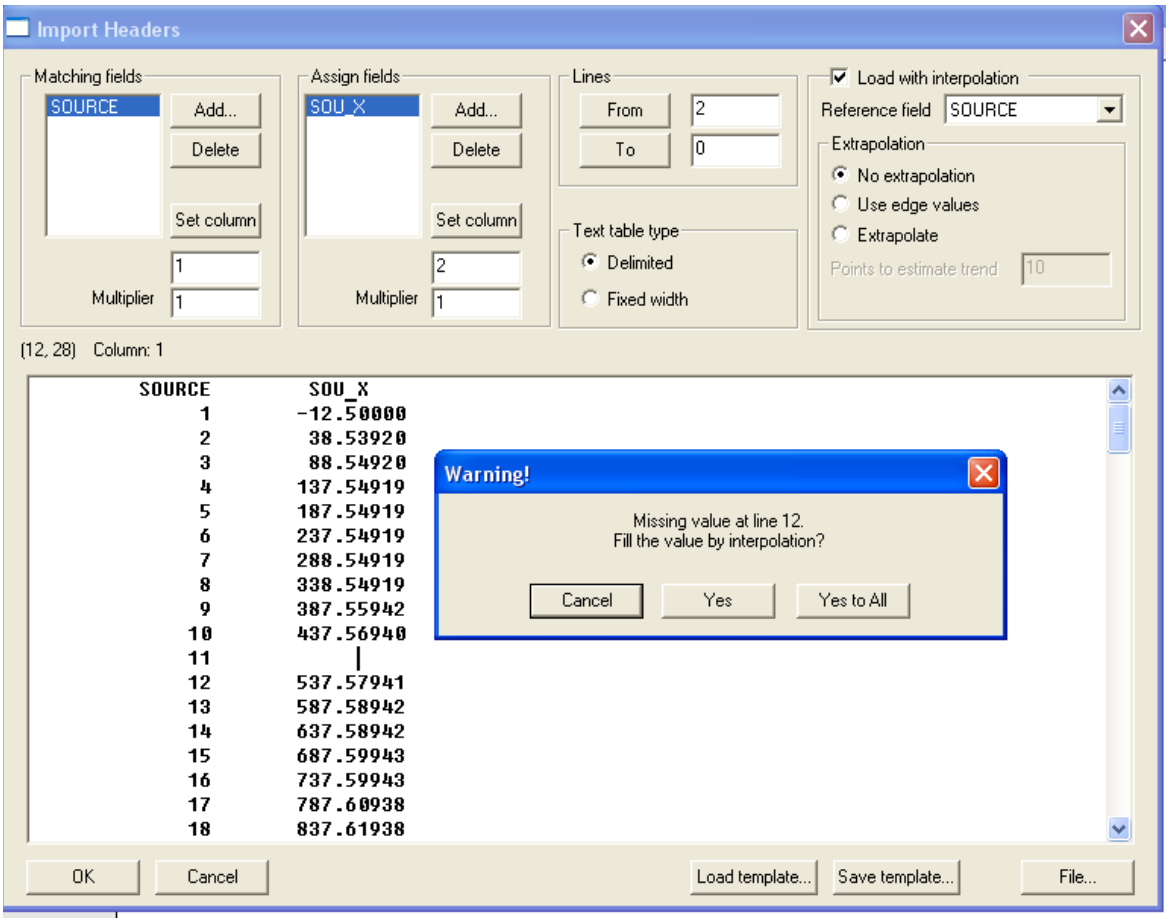
- **Multiplier** - the data from the selected column will be multiplied by this value while correlating with values of the selected header (in **Matching Fields** case) or while importing into the selected header (in **Assign Fields** case).

There is also a group of parameters **Lines** where the range of the lines in the text file should be defined. In accordance with the **Matching Fields** and **Assign Fields** fields settings the values imported will be accomplished only from the lines of the selected range. To set a number for the initial line of the range click the left mouse button (MB1) on the desired line in the text file content window and then click the **From** button. The selected line's number will appear in the **From** field. Similarly, a finite line's number can be specified in the **To** field via the **To** button. Also, the lines' numbers can be specified in the corresponding fields manually (the numerations of lines in a file starts with 0).

The **Save template** and **Load template** buttons are designed to save current import parameters in a template of the project database and to load import parameters from a previously saved template

respectively.

Load with interpolation option allows to import file with interpolation. In case of presence missing values in any part of the file – they will be linear interpolated by the header field which is set in *Matching field* block. To set the header – choose it in the **Reference field** option. By pressing OK button the following message will appear:



Cancel – procedure of importing values to headers will be canceled with returning to **Import Headers** window. By pressing **Yes** button the value, which is currently displayed on dialog box, will be interpolated. **Yes to all** — all missing values will be interpolated.

Extrapolate - extrapolates values, missing at the beginning or in the end of file:

No extrapolate – extrapolation will not be done;

Use edge values – missing values will be filled equal the last value which is present in the file

Extrapolate – extrapolation using specified trend. Trend will be calculated by number of points, selected in **Points to estimate trend** field.

Recommended procedure:

- First, click the **File...** button to select an ASCII spreadsheet file from which the values will be imported. File content will be displayed in the **Import headers** dialog box.

- Then, specify the line range from which the values will be imported. To do this, select the first line to be imported by clicking the left mouse button and then click the **From** button in the **Lines** group. The number of the line which will be imported first will be displayed in the **From** field. Click the left mouse button on the last line you would like to import and click the **To** button in the **Lines** group. The number of the last imported line will be displayed in the **To** field. (Use a scroll bar while selection if necessary). Also, the lines' numbers can be specified in the corresponding fields manually (the numeration of lines in a file starts with 0). Line range specification is compulsory even when all lines from the file are imported!

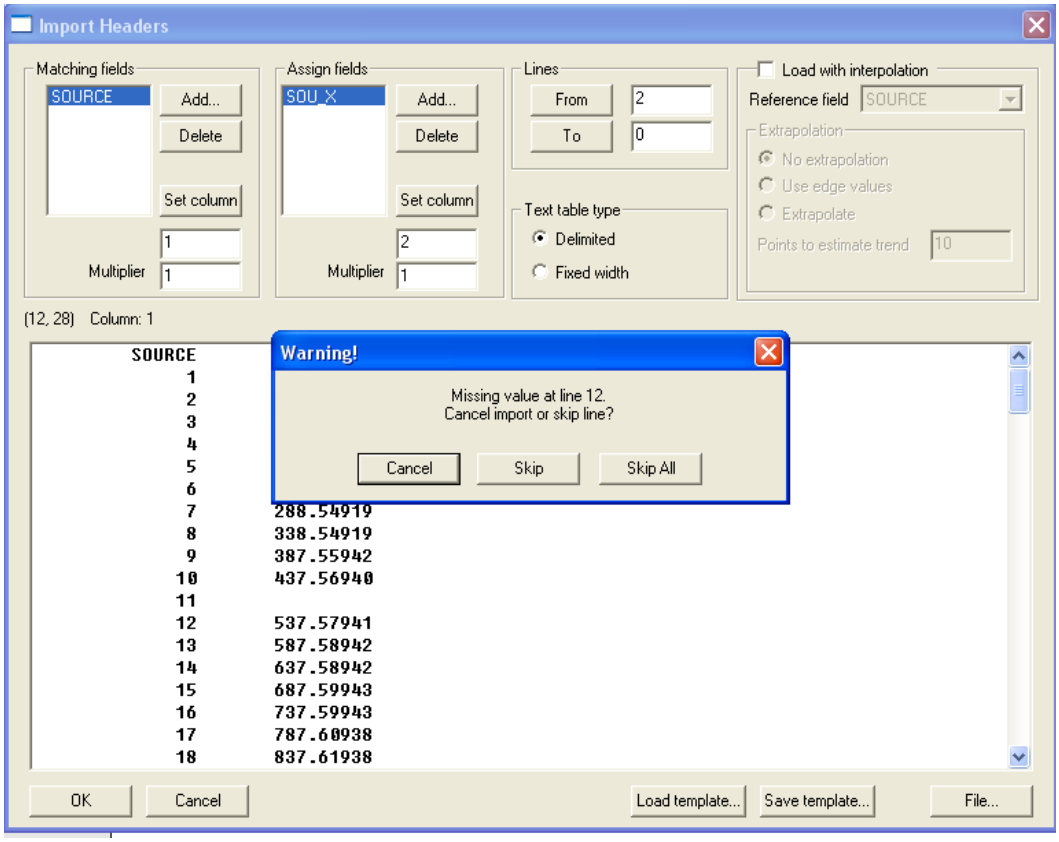
- Use **Add...** button to select the headers from the list of the headers for the **Assign fields** and **Matching fields** fields.

- Correlate selected headers with columns in the file. To do this, click the left mouse button (**MB1**) on the header name in the **Assign fields** and **Matching fields** fields, then click on the column with displayed text that corresponds to the selected header. After that, click the **Column** button and the column number will appear in the **Column** field. Set **Multiplier** value for the selected header.

- When all parameters are set, click the **OK** button. Values import will be accomplished. The header field values, where the values have been imported to (i.e. fields from the **Assign fields** list), will appear as separate columns in the **Geometry Spreadsheet** window.

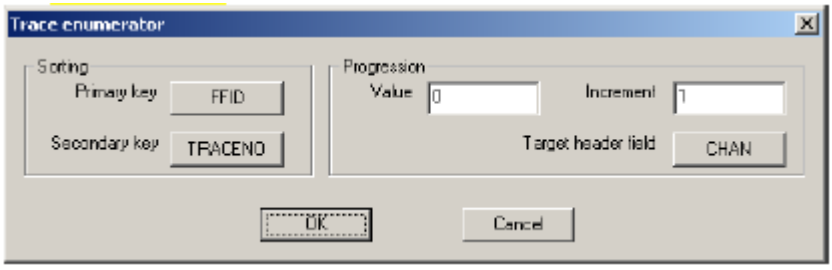
- In case of missing values in the file in any column, warning message with the position will appear. Press **Cancel** to stop importing process and return to **Import headers** window. While

pressing Skip button current value will be skipped without changing in the header (if Load with interpolation function is disabled). Press **Skip all** button to skip all missing values in file (if Load with interpolation function is disabled).



Trace Enumerator tool

A tool dialog box appears when you choose the command **Tools/Trace Enumerator** from the menu. This dialog box allows you to sort traces by two header fields and then to assign a continuous numbering index to them, the values of which are entered in the third header field.



Primary key (header field that will be used as a primary sort key) and **Secondary key** (header field that will be used as a secondary sort key) are set in the **Sorting** field.

The progression parameters for continuous numbering—**Value** and **Increment**—are set in the field **Progression** where you choose the field to which the acquired values will be written—**Target header field**.

Example of usage:

For instance, if the data include two traces with one FFID value, differing by TRACENO values, after using the Tool with parameters which are represented on the figure, the values of the field CHAN can be:

FFID TRACENO CHAN

1 1 0

1 2 1

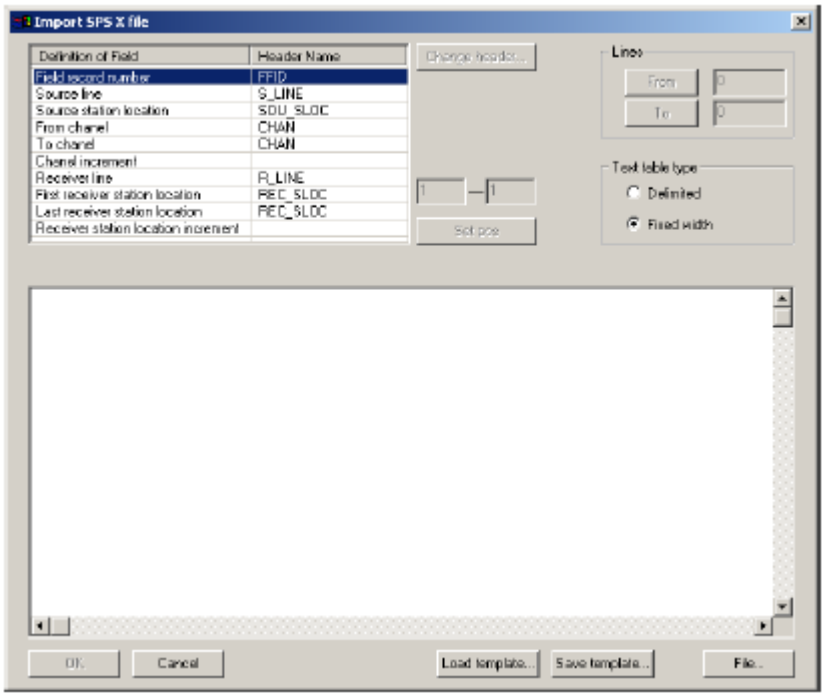
2 1 3

2 2 4

Import of SPS-X files

To import SPS-X files use the menu command **Tools/Import SPS-X.**

The following dialog import window will open:



Recommended procedure:

- Choose the SPS-X file using the **File…**button. The file content will be reflected in the lower part of the window.

- Please indicate the lines of the file to load with the number of the starting line in the **From** field and the number of the ending line in the **To** field within the **Lines** section of the parameters**.** You can also position the mouse cursor on the required line and click the **From** or **To** button—the line number will be automatically reflected in the corresponding field.

- Choose the mode of line separation into separate fields **Text table type**

Delimited by columns: the fields in line will be separated by a space.

Fixed width by positions: the fields in line will have a distinct position relative to the start of the line and a fixed width.

- Set matching between the columns (positions) in file and header fields to which the data are loaded.

The matching is carried out by the fields *Field record number* and *Chanel*. The loading is performed to the *Source line*, *Receiver line*, *Source station location, and Receiver station location*.

First of all, when you load a line you can determine the *Field record number*, corresponding to the line. Then, the channel numbers are determined for this record number using the fields *From chanel*, *To chanel*, and *Chanel increment*. Furthermore, the values are loaded to the indicated headers for each trace from the fields *Source line*, *Source station location*, *Receiver line* (pairs *record number - chanel*), described by the given line.

As well, the value of *Receiver station location* is loaded for each trace according to the fields *First receiver station location*, *Last receiver station location*, and *Receiver station location increment*.

You should set the name of a specific trace header for each field that matches.

The current header names for each field (column **Description of Field** in the list of the upper left part of the window) are visualized in the right part of the list (**Header Name** column). To change the header corresponding to any field, double click on its name with the left mouse button or use the **Change header** button.

In order to set the column (position) in the file corresponding to a field:

- Select the field in the list using the mouse.

- In delimited mode, indicate the column number or point the mouse cursor to the required column and click the **Set column** button.

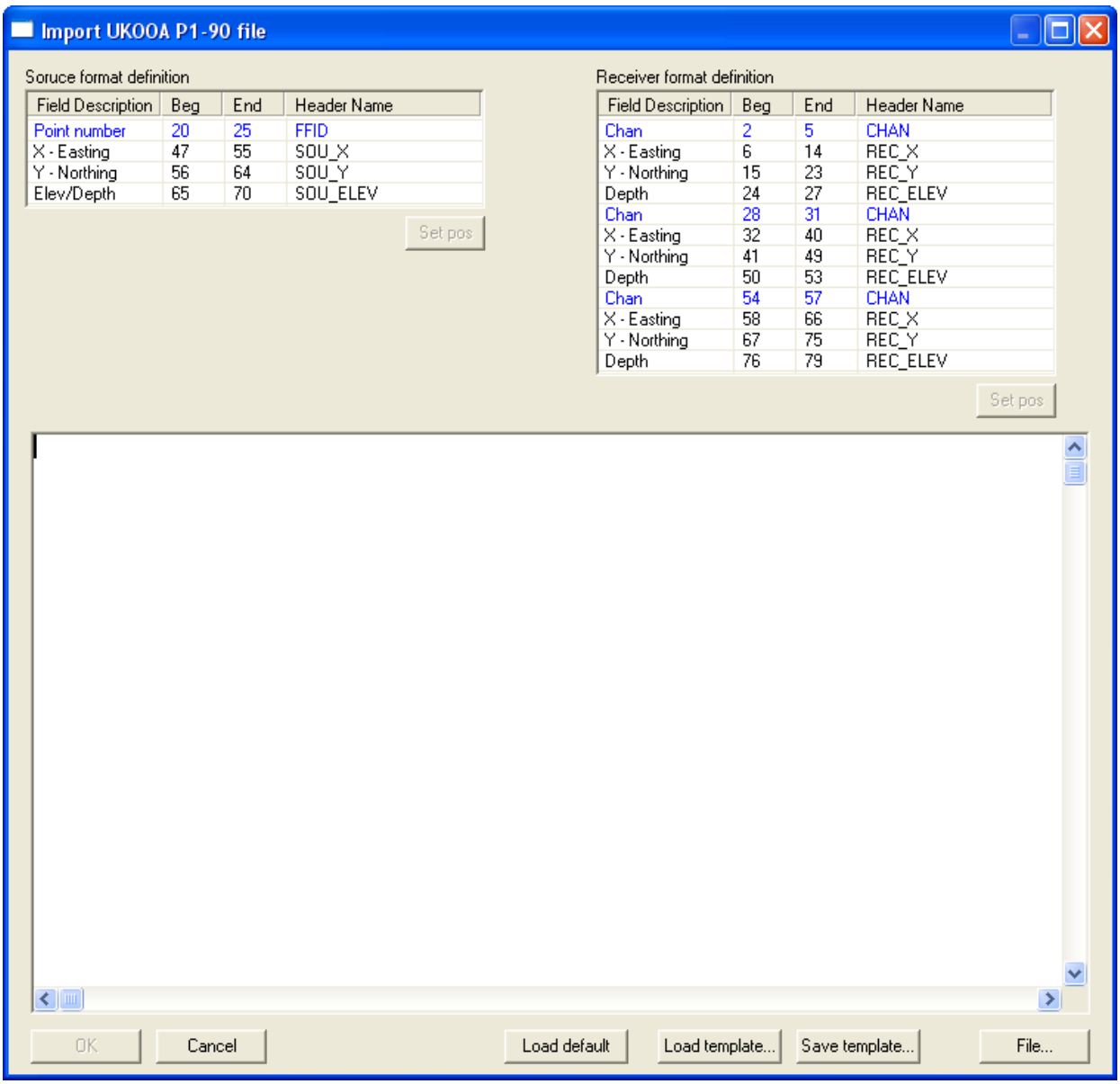
- In fixed width mode indicate the values of the first and last position in the corresponding entry fields, or select the required range using the mouse and click the **Set pos** button. The current range will be visualized in red, while the range being edited is visualized in blue (their intersection will be viewed in green). The fields *Chanel increment* and *Receiver station location increment* can be filled in by two methods: from the file (**From file**), indicating the column (positions), or manually (**Manual**), setting the value in the corresponding entry field. After setting all the parameters, click the **OK** button to load data from the SPS-X file.

UKOOA p1-90 files import

UKOOA p1-90 files are imported using the **Import UKOOA p1-90** menu command.

The program imports source and receiver coordinates and elevations from the UKOOA file; the rest of the information contained in the file is ignored. Search for the values corresponding to sources and receivers is performed based on the S and R identifiers (source and receiver identifier, respectively) located at the beginning of the line in the file.

The import dialog box looks as follows:

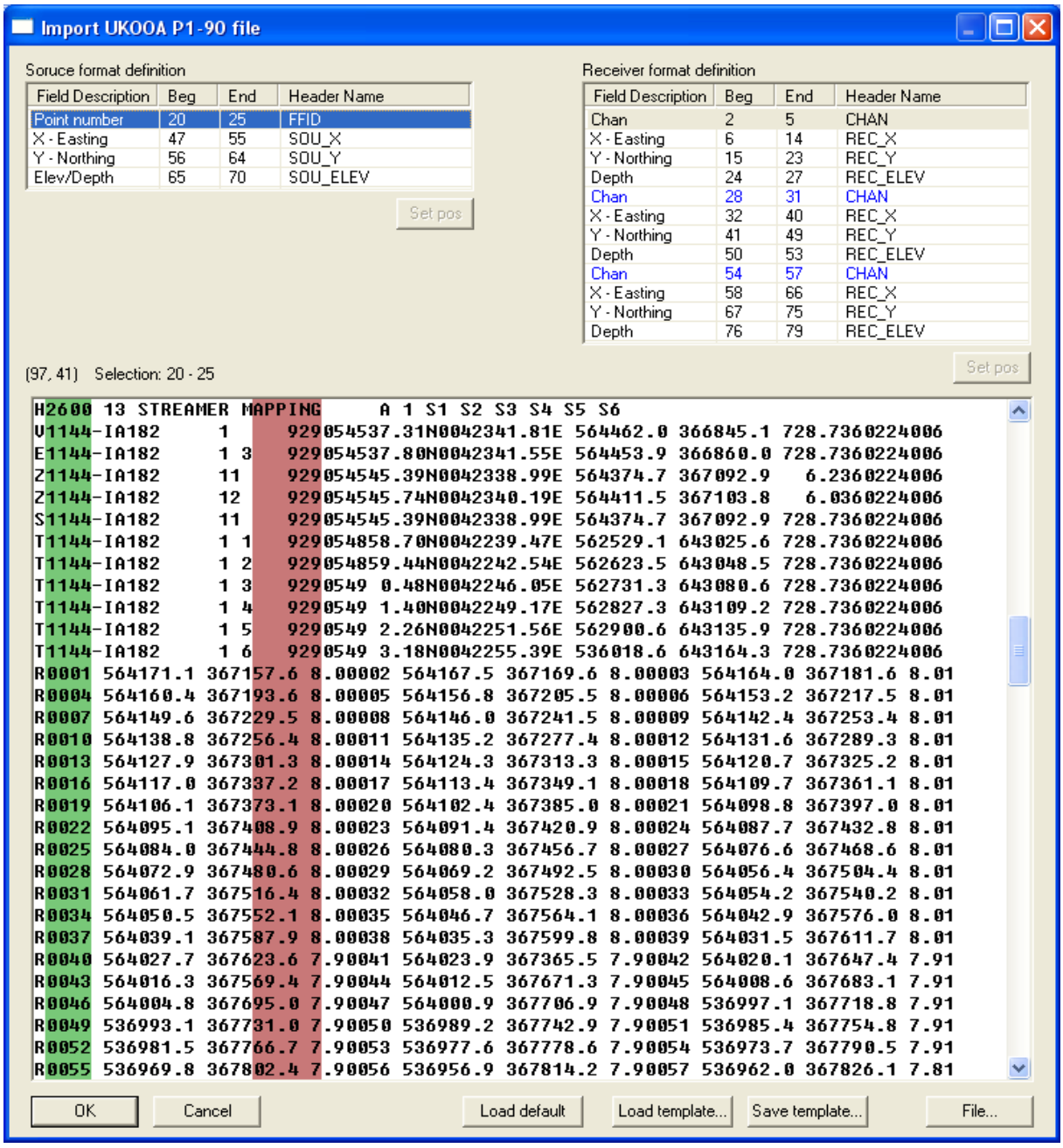

Import UKOOA P1-90 file
Soruce format definition
Field Description Beg End Header Name
Point number 20 25 FFID
X - Easting 47 55 SOU_X
Y - Northing 56 64 SOU_Y
Elev/Depth 65 70 SOU_ELEV
Set pos
Receiver format definition
Field Description Beg End Header Name
Chan 2 5 CHAN
X - Easting 6 14 REC_X
Y - Northing 15 23 REC_Y
Depth 24 27 REC_ELEV
Chan 28 31 CHAN
X - Easting 32 40 REC_X
Y - Northing 41 49 REC_Y
Depth 50 53 REC_ELEV
Chan 54 57 CHAN
X - Easting 58 66 REC_X
Y - Northing 67 75 REC_Y
Depth 76 79 REC_ELEV
Set pos
OK
Cancel
Load default
Load template...
Save template...
File...

The dialog box is divided into two main areas that control the import of headers associated with sources and receivers: **Source format definition** and **Receiver format definition**, respectively. Each import block has four columns: **Field description** – header field description, **Beg** – beginning of the column corresponding to the header value in the file being loaded, **End** – end of the column corresponding to the header value in the file being loaded, and **Header name** – header field name. Header fields to which other values are linked are shown in blue. For source coordinates this is the FFID (shot number) field, for receiver coordinates – the CHAN (channel number) field. Three repeating CHAN header, coordinate and elevation fields are specified for receivers in the geometry assignment block since the standard UKOAA p1-90 file has three columns for coordinates/elevations. By default the range of values for each of the headers is specified in accordance with the standard UKOOA p1-90 file format (format description can be found at www.seg.org, the following link is active as of September 1, 2011:

www.seg.org/documents/10161/77915/ukooa_p1_90.pdf).

Recommended procedure:

To load the file, press the **File...** button and select the file you need. Its contents will be displayed in the lower pane of the dialog box:



Left-clicking a line that contains a header field highlights the corresponding value column

(according to the standard file format). If columns with header values are invalid for some reason, they need to be redefined. To do this, select the relevant header in the **Source format definition** block, select the range of values corresponding to that header in the lower pane by clicking and holding the left mouse button, and press **Set pos**. Values in the **Receiver format definition** block are assigned in a similar manner. The **Load template...** and **Save Template** buttons allow saving and loading the current file import settings. The **Load default** button restores default UKOOA p1-90 file import settings according to the current format.

Make sure that values correspond to all headers are valid and press **OK**. The coordinate and elevation values will be written to the specified headers in the database.

RadExPro header fields

To store auxiliary information on seismic data, the **RadExPro** program uses its own set of header fields. They are kept in database files, separate from data files which provides fast access to the header fields.

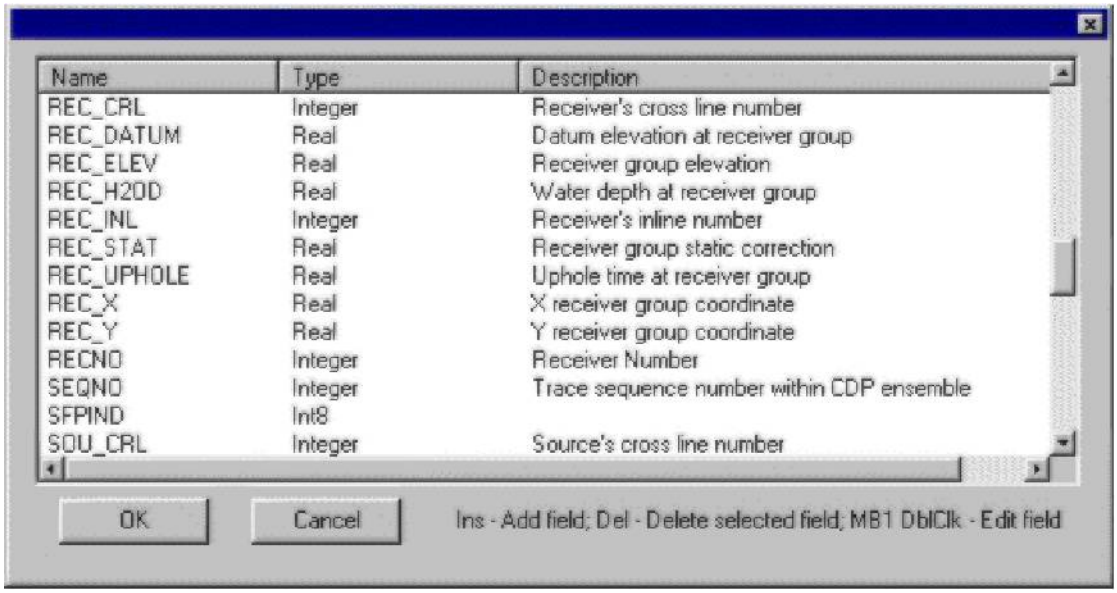
When creating a new project the set of header fields in the project is similar to trace headers in SEG-Y format. However further the header fields can be edited, i.e. you can add new fields and delete and rename the existing ones. To view and edit the active set of project header fields you should select the menu command **Database/Edit header fields...** in the main window of the program (project tree window).

In the new (or existing) header fields you can record various information. For example, seismic trace static shift or arrival time picked on the trace. You can do mathematical operations with header field values, convert them into picks, display changes of header values in different datasets, etc.

Edit Header Fields

To view and edit an active set of project header fields, select the menu command **Database/Edit header...** from the main window of the program (project tree window).

The window containing the list of headers from the active project will open:

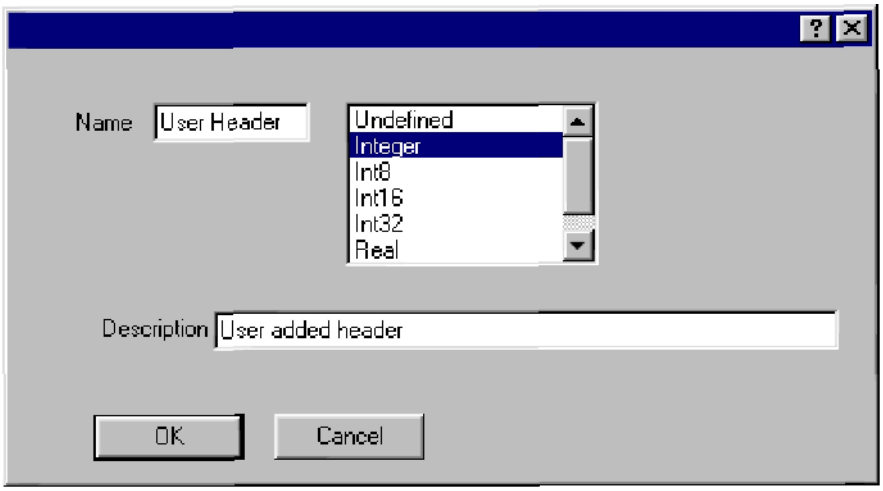


In the header list there are three fields:

- **Name:** the name of header field
- **Type:** the type of header field value
- **Description:** header description

To add a new header field, click the **Insert** button. To edit the existing header field, double-click the left mouse button (**MB1**) on the name of the required field. To delete an existing header, select it in the list with the left mouse button (**MB1**) and click the **Delete** button. After the deletion has been confirmed the field will be deleted from the project.

When adding a new header field or editing the existing one, the following dialog box will appear:



Here you can specify/change the name, format, and description of the header. Enter/change the header field name in the **Name** field.

Select the required format of the header from the list of available formats. The header field values can be stored in one of the following formats:

Integer, Int32: 32-byte integer

Int8: 8-byte integer

Int16: 16-byte integer

Real: 32-byte real with floating point

Real8: 64-byte real with floating point

In the **Description** field, specify the annotation or textual description of the header field.

After all parameters have been set, click the **OK** button. To exit without saving the changes click the **Cancel** button.

Important header fields

Below you can find a list of some most important standard **RadExPro** header fields. We advise you not to delete these fields or change their meaning.

- **DT:** sampling interval in ms
- **NUMSMP:** number of samples per trace
- **FFID:** seismogram number (shooting number)
- **SOURCE**: source point number
- **CDP:** CDP number
- **RECNO:** receiver point number
- **SOU_X, SOU_Y:** X coordinate of the source, Y coordinate of the source
- **REC_X, REC_Y:** X coordinate of the receiver, Y coordinate of the receiver
- **OFFSET:** distance between the source and the receiver
- **AOFFSET:** absolute value of the offset

- **CDP_X, CDP_Y:** X coordinate of the CDP, Y coordinate of the CDP

- **SFPIND:** identification number of dataset used while displaying objects on the map. For 3D data, it is given automatically when the file is registered in the project. In the case of 2D data, the profile number is specified in the Profile ID field in the SEG-Y Input or Add data file window.

- **SOU_INL, SOU_CRL:** the source number along the observation line, the source number across the observation line

- **REC_INL, REC_CRL:** the receiver number along the observation line, the receiver number across the observation line

- **ILINE_NO, XLINE_NO:** CDP number along the observation line, CDP number across the observation line

A correspondence of RadExPro header fields to SEG-Y standard trace header is provided in the description of the **SEG-Y Output** processing module.

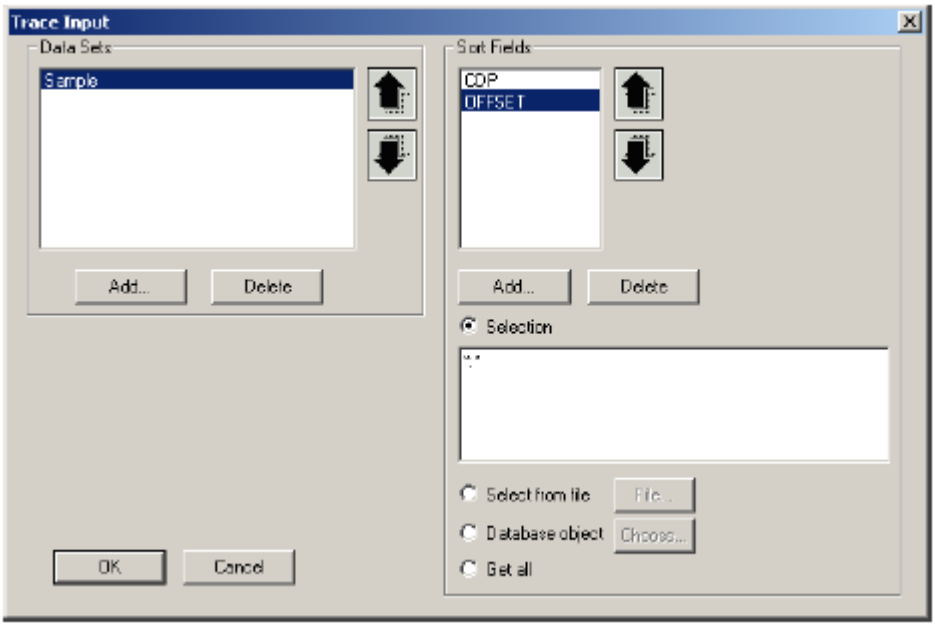
Processing modules

Data I/O (Data input/output)

Trace input

This module is designed to load seismic data files registered in the project into the flow.

When launching this module, the following window will appear:



The **Add…**button from the **Data Set** field calls a standard dialog box **Database Object Selection** (see **Database Object Selection** window) with **Choose dataset** header, where you select the needed seismic dataset. You may choose one or several data sets. The names of the chosen datasets will appear in the **Data Sets** list. You may change the position of data sets in the list using the buttons with arrows to the right of the list (while reading the unsorted data using the **Get all** option may be useful). The data sets, selected using the mouse can be removed from the field, using the **Delete** button. The **Add…**button from the **Sort Fields** field calls a standard dialog window, containing the list of header fields. You need to choose sort keys from the list, in order to perform the input data sorting. The selected keys will appear in the list **Sort Fields**. Mutual position of keys on the list can be changed using the buttons with arrows to the right of the list. You can delete selected keys from the list using the **Delete** button.

If the option **Selection** is on, the data—input into the flow—will be sorted in accordance with the sort keys that are given in the field **Sort Fields**. You should write a line with the input range for each key in the **Selection** field. A colon separates the ranges of different keys.

For example:

Let's assume that 2 sort keys are selected in the field **Sort Fields**. Then the range line in the **Selection** field will look like the following:

: - all data, sorted according to two selected keys will be input

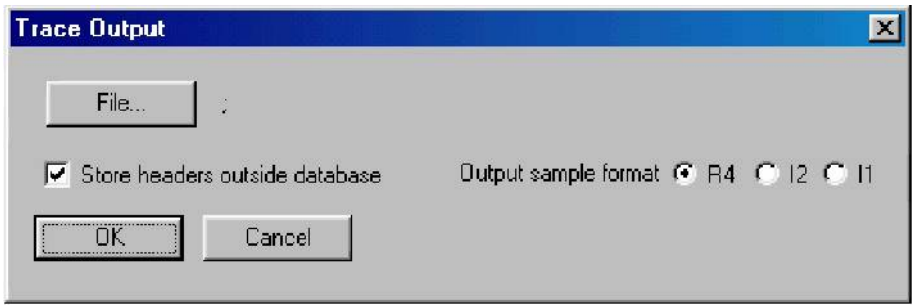
*:1000-2000(5) – the data, sorted by the second key will be input within the range of 1000 to 2000 with 5 as an increment.

The **Select from file** option loads the input string from the text file range. Click the button **File…**, choose the required file with the text *samples assignment* from the pop-up window.

The **Get all** option allows loading the whole data set in the order they were recorded without any additional sorting. You don't need to assign sort keys and ranges. When you choose this option the corresponding fields become inaccessible and their content doesn't affect the result. If the field **Data sets** contains several data sets, the first data set input in the flow will be the uppermost in the list, then the next one, etc. To assign the required input sequence, use the buttons with arrows to the right of the list **Data sets**.

Trace Output

This module is used to record the data from the flow into the program database in any of the formats: R4, I2, I1. When launching this module the following window will appear:



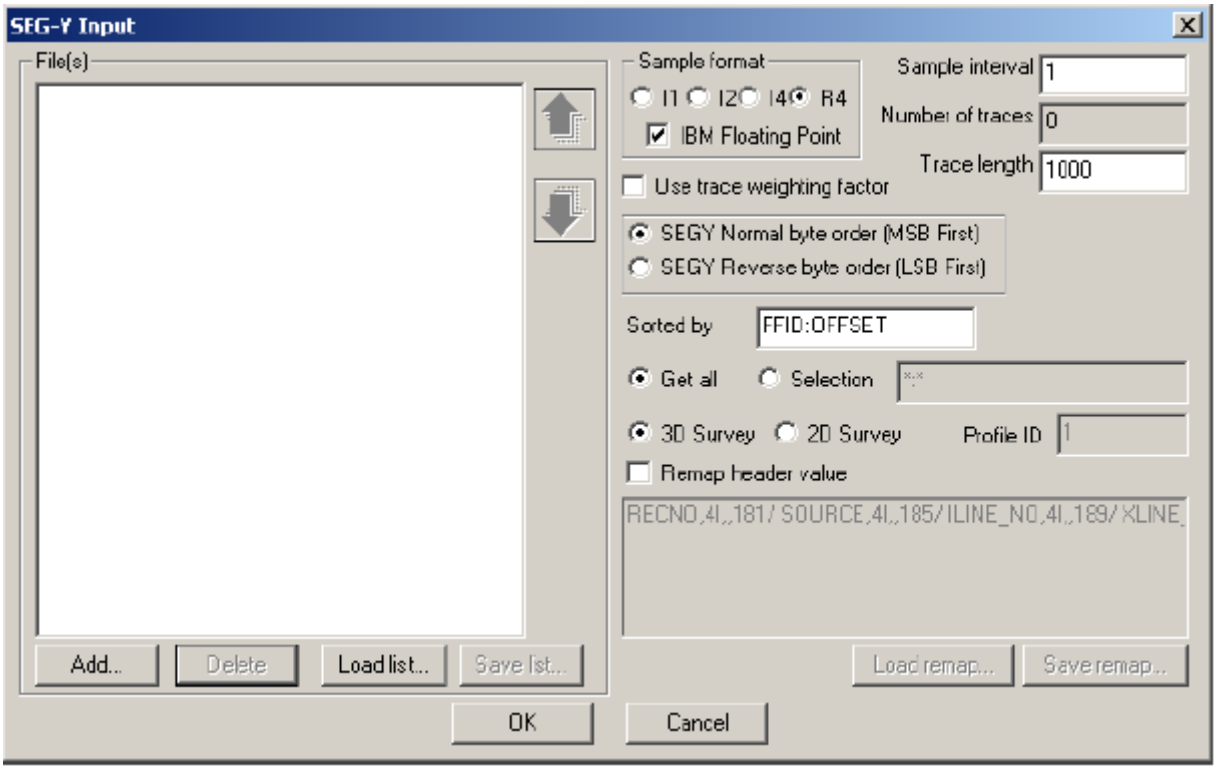
When clicking the **File...** button the **Select Dataset** dialog box will appear. Here, in the **Location** field, define the path to where the file to be created will be stored. The name for the file that will be created should be entered into **Object name** field or, if the user wants to save the file under an old name, it should be selected from the list in the **Objects** field.

Activation of the **Store headers outside database** option saves the headers outside the database (this is the recommended method).

In the **Output sample format** field the user can choose the desired format for saving the data of the file under creation.

SEG-Y input

This module is designed to input external (with respect to the project)files in SEG_Y format,, into the flow. When this module is activated the following window appears:



To choose input files click **Add...**. In the dialog box you may choose one or several files from one directory (to choose several files use the *Ctrl* and *Shift* buttons). After you have chosen the files, their names will appear in the **File(s)** field. When you choose several files check the correctness of their mutual position in the list: the uppermost file from the list will be loaded to the flow first, then the next one, and so forth from top to bottom. If you need, you may correct the mutual position of the files in the list. Select the file for which the position is to be changed using the mouse and, using the corresponding buttons with arrows to the right of the list, move it up or down.

You may delete one or several selected files from the list using the **Delete** button. The other method is to save the current file list as a text file on a disk (**Save list...**) and to load the list of files (**Load list...**).

The following parameters will be automatically set after choosing SEG-Y files, from the headers of the first file from the list. However, if you need you may change the values of parameters manually. Indicate the format to read the samples from the input file in the field **Sample format**. Here:

I1 – 8-bit integer

I2 – 16-bit integer

I4 – 32-bit integer

R4 – 32-bit floating point real, or IBM floating point, or IEEE as a function of option **IBM Floating Point**

Options **SEG-Y Normal/Reverse byte order (MSB first)/(LSB first)** set normal and reverse byte order in the word.

After choosing SEG-Y files, the sampling interval in ms will be reflected in the field **Sample interval**, the trace length in file, in samples, will be shown in the field **Trace length**. The values can be changed by the user and any change influences the process of file reading. However the values taken from the traces of the input file will still be in the header fields.

After opening the file, the number of traces in the **Number of traces** field in the selected SEG-Y file will be displayed. This value is calculated depending on file size and trace length. If the user has changed the trace length in the **Trace length** field, in order to recalculate the trace number in the file, click the **OK** button and open the dialog box of the module in the flow again. A new value corresponding to the changed trace length will appear in the **Number of traces** field.

The option **Use trace weighting factor** indicates the necessity to take into account the normalizing coefficient during amplitude recovery, recorded in bytes 169–170 of the trace's header (see module **SEG-Y Output** description). The option is checked by default, however in cases, when the bytes are filled in incorrectly (for example, contain “trash”) this option needs to be toggled off. The sort keys of the input data are indicated in the **Sorted by** field (by header names). The change of the keys doesn’t have any impact on the real input data sorting, however it allows making the set by values’ ranges of the indicated keys using the **Selection** option.

The options **Get all/Selection** gets all the data or a part of the data confined by primary and secondary keys, indicated in the field **Sorted by**. If we input a part of the data, then the limitations on the primary and the secondary keys are indicated in the **Selection** field.

For example:

: - all data will be input,*: **1000–2000(5)** - according to the secondary key the data will be input in the range of 1000 to 2000 with step 5.

Depending on the type of input data (3D/2D) the **3D Survey/2D Survey** options, respectively, should be selected. If the input data are 2D data **(2D Survey** option is selected), then you should indicate the unique profile number in the **Profile ID** field.

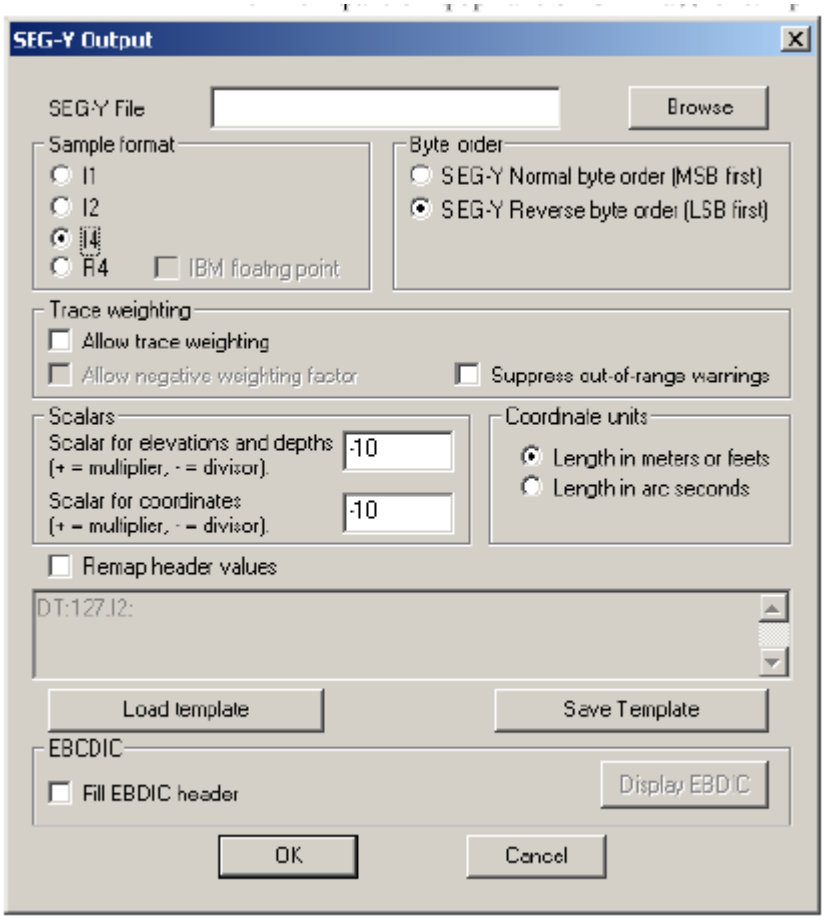
The **Remap header values** option (see **Add data file to project** chapter) sets the format of the remap.

The **Save template...** button saves the active remap into the database.

The **Load template...** button loads the remap previously saved in the database.

SEG-Y output

This module serves to save the (processed) data from the flow, into an external file in SEG-Y format on disk. When this module is activated the following window appears:



To enter the name of the input SEG-Y file where the data will be saved, click the **Browse...** button. After the file has been chosen, its name and path will be displayed in the **File** field. You can enter the output file name in the **File** field manually.

The format in which the samples will be recorded into the file should be defined in the **Sample format** field. Here:

I1 – 8-byte integer

I2 – 16-byte integer I4 – 32-byte integer

R4 – 32-byte real with floating point, or IBM floating point, or IEEE depending on **IBM Floating Point** option state

The **SEG-Y Normal/Reverse byte order (MSB first)/(LSB first)** options assign normal or reverse byte order in a word, respectively.

The Trace weighting field saves data in integer format without loss of raw dynamic range. The options of this field become accessible, if one of the integer formats is chosen in the field **Sample format** (**I1**, **I2,** or **I4**).

According to SEG-Y format specifications, N value can possess the values 0, 1, … 32767. However, in case the dynamic range of data exceeds that of the chosen integer format, there is an alternative option of using negative values of N. To permit the usage of negative N values, check the option **Allow negative weighting factor** (this option is accessible when you toggle on **Allow trace weighting**).

If you don't use **Trace weighting**, or if you use only positive N values, in cases where the amplitude values go beyond the dynamic range of the record format, the following warning appears during the run of the module:

To toggle off the warning, toggle on the option **Suppress out of range warnings**.

The **Remap header values** option allows format remapping while SEG-Y-file recording (see **Add data file** chapter).

The **Save template** and **Load template** buttons save the active format remap into the project database and loads a previously saved remap from the database, respectively.

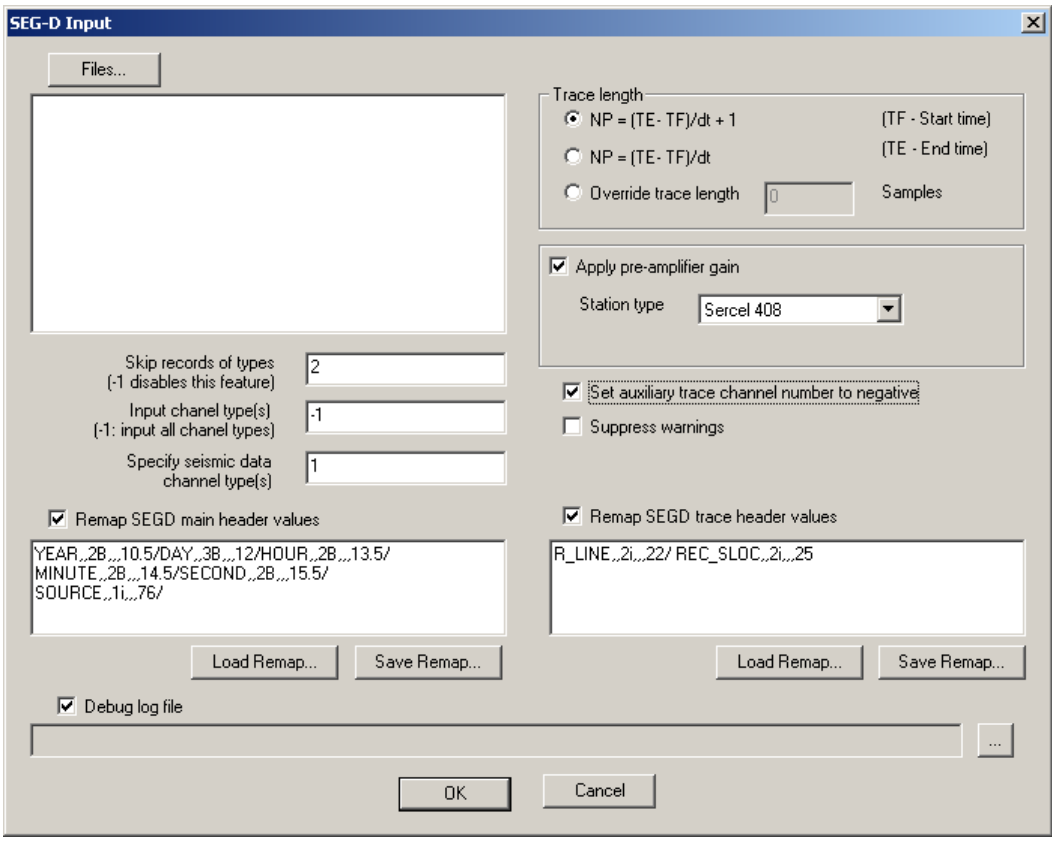
RadExPro Plus header fields corresponding to SEG-Y trace header by default

RadExPro Plus header field	Byte numbers of SEG-Y trace header	Comment
TRACENO	1–4	Sequence number of the trace
FFID	9–12	Original field record number
CHAN	13–16	Channel number (trace number within the original field record)
SOURCE	17–20	Source point number
CDP	21–24	CDP ensemble number
SEQNO	25–28	Trace number within the CDP ensemble
TRC_TYPE	29–30	Trace identification code
STACKCNT	31–32	Number of vertically summed traces yielding this trace
TRFOLD	33–34	Number of horizontally stacked traces yielding this trace
OFFSET	37–40	Offset (distance from center of the source point to the center of the receiver group)
REC_ELEV	41–44	Receiver group elevation
SOU_ELEV	45–48	Surface elevation at source
DEPTH	49–52	Source depth below surface
REC_DATUM	53–56	Datum elevation at receiver group.
SOU_DATUM	57–60	Datum elevation at source.
SOU_H2OD	61–64	Water depth at source.
REC_H2OD	65–68	Water depth at receiver group.
SOU_X	73–76	Source X-coordinate
SOU_Y	77–80	Source Y- coordinate
REC_X	81–84	Receiver group X- coordinate
REC_Y	85–88	Receiver group Y- coordinate
UPHOLE	95–96	Uphole time at source in ms
REC_UPHOLE	97–98	Uphole time at receiver group in ms
SOU_STAT	99–100	Source static correction in ms
REC_STAT	101–102	Receiver group static correction in ms
TOT_STAT	103–104	Total static correction in ms
TLIVE_S	111–112	Mute time — Start time in milliseconds.
TFULL_S	113–114	Mute time — End time in milliseconds.
NUMSMP	115–116	Number of samples is trace (**RadExPro** Plus specifies: one value for all traces from the internal parameters of the data frame!)
DT	117–118	Sample interval in microseconds (**RadExPro** Plus specific: one value for all traces from the internal parameters of the data frame!)

IGAIN	119–120	Gain type code of field instruments
PREAMP	121–122	Instrument gain constant (dB)
EARLYG	123–124	Instrument early or initial gain (dB)
COR_FLAG	125–126	Correlation flag (1=yes, 2=no)
SWEEPFREQSTART	127–128	Sweep frequency at start (Hz)
SWEEPFREQEND	129–130	Sweep frequency at end (Hz)
SWEEPLEN	131–132	Sweep length in ms
SWEEPTYPE	133–134	Sweep type code
SWEEPTAPSTART	135–136	Sweep trace taper length at start in milliseconds
SWEEPTAPEND	137–138	Sweep trace taper length at end in milliseconds
SWEEPTAPCODE	139–140	Taper type code
AAXFILT	141–142	Alias filter frequency (Hz)
AAXSLOP	143–144	Alias filter slope (dB/octave)
FREQXN	145–146	Notch filter frequency (Hz)
FXNSLOP	147–148	Notch filter slope (dB/octave).
FREQXL	149–150	Low cut frequency in Hz
FREQXH	151–152	High-cut frequency in Hz
FXLSLOP	147–148	Low-cut slope in dB/oct
FXHSLOP	155–156	High-cut slope in dB/oct
--	169–170	Trace weighting factor defined as $1/2^N$, where N =0,1, …,32767 for integer formats (**RadExPro** Plus specific: negative integer values are allowed if the option **Allow negative weighting factor** is toggled on!)

SEG-D Input

This module is designed to input external (relative to the project) data from disk files in SEG-D format into the flow. The parameters dialog looks like the following:



A method of calculating number of samples

As there are different ways to calculate the number of trace samples for different seismic stations, you should indicate the calculation method in the parameter section **Trace length**:

- **NP = (TE-TF)/dt + 1 –** if a zero sample corresponds to the start time (TF from Channel Set Descriptor), while the last sample corresponds to the end time (TE from Channel Set Descriptor)

- **NP = (TE-TF)/dt –** if there is no information for the end time (TE) in data

- **Override trace length** – the given mode allows the user to specify the number of trace samples in manual mode

Preamplifier gain

The **Apply pre-amplifier gain** parameter allows choosing **Station Type.** The station type selected

affects the position in SEG-D file from where the descale multiplier (MP-factor) will be read.

Skip records of types

If you specify the record type in the field **Skip records of types,** the records of the indicated type will not be loaded by the module. If you indicate -1 in this field, the records of any type will be loaded. If you have to indicate several record types, you should use a colon as the separator

Input channel type(s)

The field **Input channel type(s)** indicates the channel type of traces input into the flow. Different types of channels are separated by a colon. If you indicate -1 in this field, all channels will be input into the flow.

Example: 1:2 – the traces of channels 1 and 2 are input.

Specify seismic data channel type(s)

The field **Specify seismic data channel type(s)** indicates the types of channels containing seismic data. It is used along with **Set auxiliary channel number to negative** parameter. When it is set, the values of the channel numbers, which are not considered seismic channels, are negative.

Suppress warnings

Exceptional cases can sometimes arise when reading data from files. If the cause of such a situation is not critical, the module can process it in a special way (for instance, in the case of a mismatch of sample numbers in different sets of channels) or proceed to read from another shot/file. The notification to a user is made in the form of a small message window and the module stops operation and waits until the user presses **OK**. Often that kind of behavior is unwanted, and to suppress the notification on the screen you are recommended to tick the flag **Suppress warnings**. In this case all messages will be written to a log-file, if it is indicated.

Remap SEGD main header values.

Allows specifying **Remap** of shot header into the **RadExPro** headers of traces being loaded, i.e., you can save a trace header with the indication of the number format and its position (starting position + increment on channels set) in the file header. If the indicated header is not registered in the list, the remap cannot come into effect. The number format must be one of the following: 1I, 2I, 4I, #B, #C, 4RIBM, 4RIEEE, 8RIEEE. The line of header remap consists of six fields and a symbol '/' at the end as a record separator, relating to different headers. The fields within the record are separated by commas.

For example:

YEAR,Year of shot, 2B,,,10.5 / SOURCE,,1i,,,76/

Field 1: YEAR – the name of the registered **RadExPro** trace header field

Field 2: Year of shot – description of header field (used only for record descriptive purposes), can be omitted.

Field 3: 2B – format of value in the header SEGD. This field can be one of the following: 1I (1-byte integer), 2I (2-byte integer), 4I (4-byte integer), #B (1–9 BCD numbers), #C (1–9 numbers in symbolic notation), 4RIBM (4-byte IBM floating point), 4RIEEE (4-byte IEEE floating point), 8RIEEE (8-byte IEEE double precision floating point). Number format BCD as well as numbers in symbolic notation are indicated in the form: 'number B', for example, 3B for three numbers BCD or 4C for four numbers in symbolic notation.

Field 4: 1:2:3 or flex – field to control channel sets. If this field is void, it means that all channel sets are selected. If this field contains set numbers (1:2:3), to get a position to read the value from, the starting position (field 6) is incremented by product (set number)*(increment (Field 5)). The set numbers should be separated by colons. The headers are written only for traces that belong to the indicated sets. This header will not be rewritten for traces from the other channel sets.

byte position = start byte position + (channel set – 1)*increment

If the field contains the identifier flex, in order to get a position to read the value from, the start position is incremented by the product (overall number of channel sets) * (increment).

byte position = start byte position + (total channel sets)*increment

Field 5: Increment - used together with field 4 to ensure flexibility in indication of the read value position. If there is a number in this field the start position (Field 5) is incremented by the indicated number of bytes for each channels' set or for all sets of channels. If the field is void, the increment is equal to zero. If the input number is not a BCD number, the increment should not be multiple of the byte, as the byte contains two BCD numbers.

For example: K1,,3B,,1.5, 0.5/

Here the value for traces from the first set are BCD numbers 1–3, and for the traces of the second set BCD numbers 4–6.

Field 6: 10.5 – is the starting position in SEGD header. If the input number is a BCD number, the increment can be in full or half bytes (nibbles). In that case, in order to read the first BCD digit from the first nibble of the byte you should indicate the position that is half a byte less than the byte number. For example, the first nibble of byte 3, has position 2.5, while the second nibble of the byte 3 has position 3.

Remap SEGD trace header values

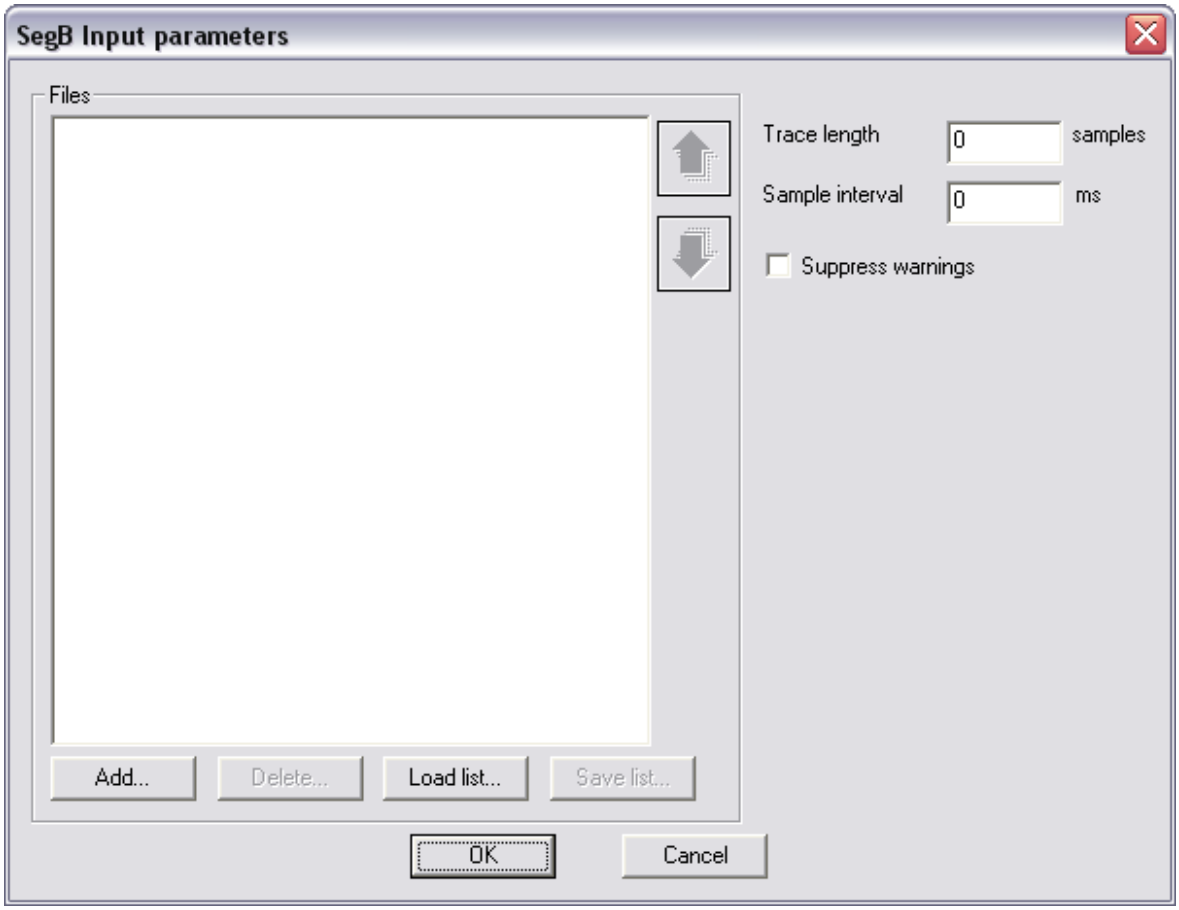
Allows remapping (Remap) SEGD trace headers into **RadExPro** trace headers. The remap line format is identical to the case of shot header; the only exception is that field 4 and Field 5 should be void.

Log file

All messages, as well as other useful information on the file being loaded (for instance the main shot header, channel set headers, and trace headers) can be saved to a log-file. To do so you can switch on the option **Debug log file** and indicate the text file name, where you prefer to output the report on the job run. This possibility can be used for error diagnostics and exception cases.

SEG-B Input

This module is designed to input external (with respect to the project) files in SEG-B format into the flow. The parameter dialog of the module is shown on the following picture:



Click **Add...** button to choose input files. In the dialog box you may choose one or several files from one directory (to choose several files use the *Ctrl* and *Shift* buttons). After you have chosen the files, their names will appear in the **File(s)** field. When you choose several files check the correctness of their mutual position in the list: the uppermost file from the list will be loaded to the flow first, then the next one, and so forth downwards. If you need, you may correct the mutual position of the files in the list. Using the mouse select the file (or several files) for which the position is to be changed and, using the corresponding buttons with arrows to the right of the list, move the selected file (or files) up or down.

You may delete one or several selected files from the list using the **Delete** button.

It is also possible to save the current file list as an ASCII text file on a disk (**Save list...**) and to load the list of files (**Load list...**) from an ASCII text file.

IMPORTANT! All input files in the list must have the same trace lenghts and sample intervals!

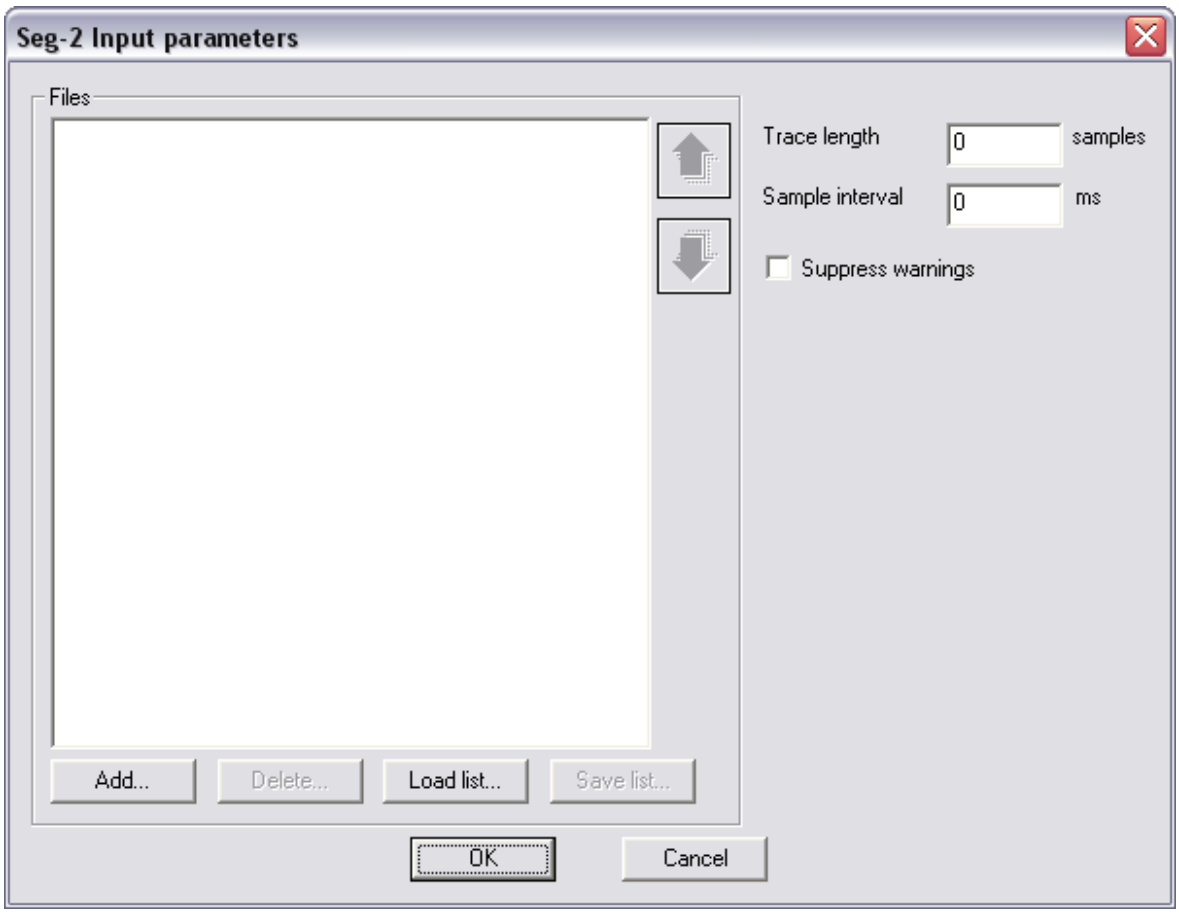
When the files are selected the **Trace Length** (in samples) and and **Sample Interval** (in ms) are detected from the headers of the last file in the list. You can check the values for any other file from the list by double-clicking on its name.

If the trace length and/or sample interval are determined incorrectly, you can change the values manually. When the traces are read from the files, the values of trace length and sample intervals are taken from the corresponding dialog fields, not from the file headers.

Exceptional cases can sometimes arise when reading the data, e.g. sample interval indicated in the file headers may differ from the value indicated in the **Sample Interval** field of the dialog that will be actually used. Then a warning message box is demonstrated and the module operation stops waiting until the user clicks the **OK** button. Often this kind of behavior is undesirable – set the **Suppress warnings** not to see the warning messages.

SEG-2 Input

This module is designed to input external (with respect to the project) files in SEG-2 format into the flow. The parameter dialog of the module is shown on the following picture:



Click **Add...** button to choose input files. In the dialog box you may choose one or several files from one directory (to choose several files use the *Ctrl* and *Shift* buttons). After you have chosen the files, their names will appear in the **File(s)** field. When you choose several files check the correctness of their mutual position in the list: the uppermost file from the list will be loaded to the flow first, then the next one, and so forth downwards. If you need, you may correct the mutual position of the files in the list. Using the mouse select the file (or several files) for which the position is to be changed and, using the corresponding buttons with arrows to the right of the list, move the selected file (or files) up or down.

You may delete one or several selected files from the list using the **Delete** button.

It is also possible to save the current file list as an ASCII text file on a disk (**Save list...**) and to load the list of files (**Load list...**) from an ASCII text file.

IMPORTANT! All input files in the list must have the same trace lenghts and sample intervals!

When the files are selected the **Trace Length** (in samples) and and **Sample Interval** (in ms) are

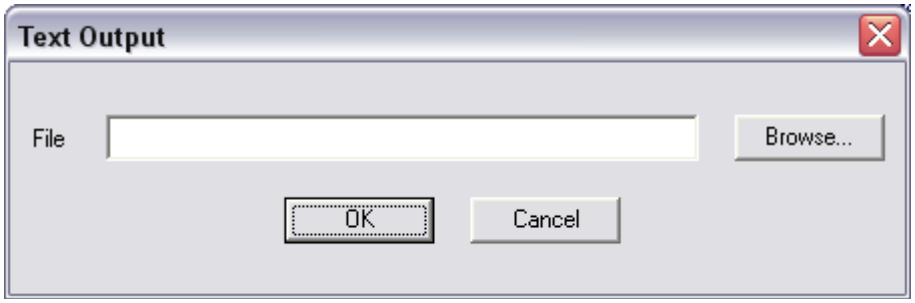
detected from the headers of the last file in the list. You can check the values for any other file from the list by double-clicking on its name.

If the trace length and/or sample interval are determined incorrectly, you can change the values manually. When the traces are read from the files, the values of trace length and sample intervals are taken from the corresponding dialog fields, not from the file headers.

Exceptional cases can sometimes arise when reading the data, e.g. sample interval indicated in the file headers may differ from the value indicated in the **Sample Interval** field of the dialog that will be actually used. Then a warning message box is demonstrated and the module operation stops waiting until the user clicks the **OK** button. Often this kind of behavior is undesirable – set the **Suppress warnings** not to see the warning messages.

Text Output

The module is designed to output the data from the flow to an external ASCII file. The parameter dialog of the module looks as following:



Click the **Browse** button to select an output file name, or type it manually to the **File** field.

As a result of the module operation, an ASCII file of the following type will be generated:

T	Tr0000	Tr0001	Tr0002	Tr0003	Tr0004....
00.0	-90.81	18.28	0.00	-0.16	3.44
02.0	-41.16	5.91	-0.74	-0.38	2.27
04.0	6.24	-21.21	-1.22	-0.39	1.27

06.0 50.28 -54.04 -0.84 -0.25 1.93

.....

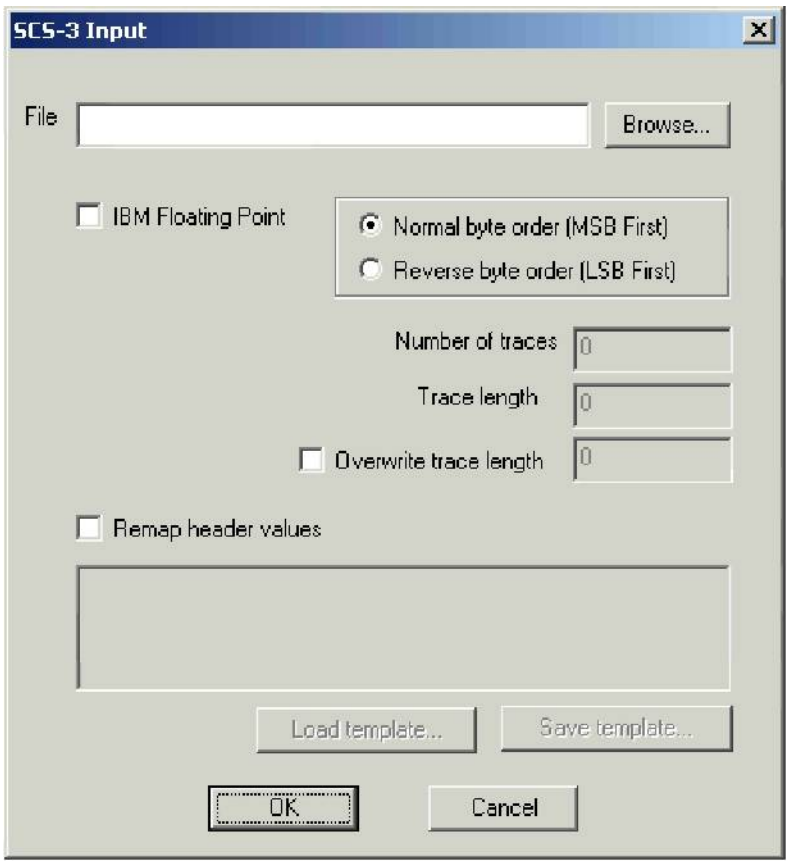
The first column (T) contains the two-way time values corresponding to each of the samples of the seismic traces.

The following columns are the traces. They contain the amplitudes of the samples. The column names (Tr0000, Tr0001, Tr0002, etc.) contain the trace sequential numbers (0-based).

IMPORTANT! When saving to ASCII the module truncates the amplitudes to 2 decimal digits. This may lead to the dynamic range loss (e.g. if all the amplitudes in the dataset varies within the range of -0.009 - +0.009 the output file will contain only zeros). In such a situation it is recommended that the *Trace Math* routine is placed in the flow in front of the *Trace Output* to multiply all amplitudes by a constant.

SCS-3 Input

This module is designed to input the external (with respect to the project) SCS-3 files into the flow. When this module is activated the following window appears:



To set the name for the input of the SCS-3-file, click the **Browse...** button. After the file has been chosen, its name and path will appear in the **File** field. You can also enter the input file name manually in the **File** field.

The **Normal/Reverse byte order (MSB first)/(LSB first)** options assign normal or reverse byte order in a word, respectively.

The **IBM Floating Point** option defines the format for the real numbers (IBM floating point or IEEE).

The **Number of traces** field reflects the number of traces in the selected file.

In the **Trace length** field the number of trace counts in the selected file is displayed.

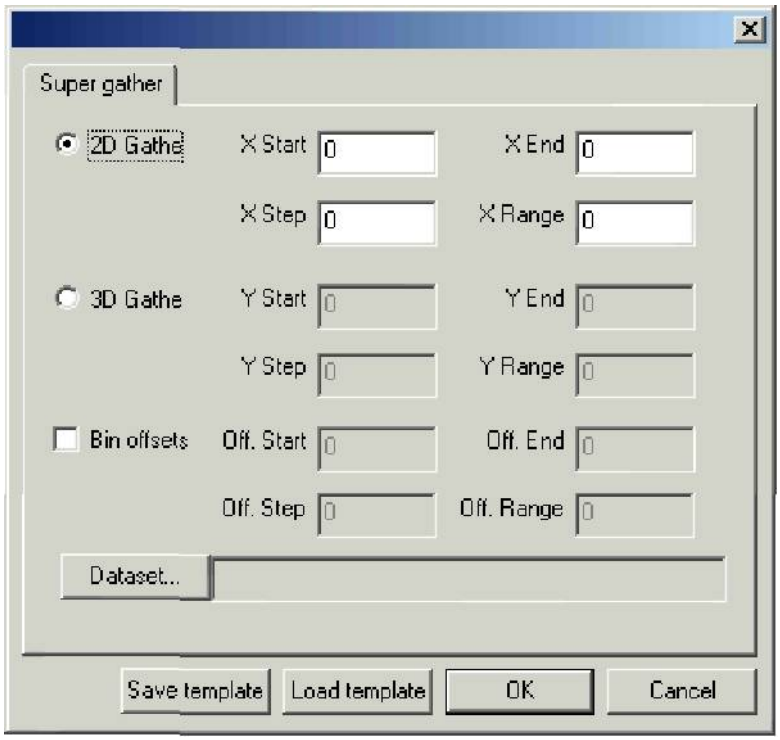
The **Override Trace Length** flag allows access to the value in the **Trace length** window. The number of trace samples defined by the user will prevail over the number of samples calculated from the header fields of the first trace.

The **Remap header values** (see **Add data file** chapter) option allows format remap setting. The **Load template...** button loads the remap previously saved in the database. The **Save template...** button saves the current remap into database.

Super Gather

This module creates and inserts into the flow, the trace sets (supergathers) composed of several CDP gathers (in 2D case) or of several In Lines-Cross Lines (in 3D case), breaking the created set into trace groups with a specified constant offset step (binning), subsuming the binned traces within the set.

When this module is activated, the following window appears:



Depending on whether the data are 2D or 3D the **2D Gather** or **3D Gather** option, is chosen.

If **2D Gather** option is chosen, then:

- In the **X Start** field, set the number for the CDP start point for the sets' creation, in the X **End** field, specify the end point.

- In the **X Step** field, specify the the interval (in CDP numbers) between the neighboring sets.

- In the **X Range** field, indicate the number of CDP points to be included in one set.

If the **3D Gather** option is chosen, then:

- In the **X Start** field, specify the start In-Line for set's creation, in X **End** field, specify the end In-Line.

- In the **X Step** field, specify the interval in In-Line numbers between the neighboring sets.

- In the **X Range** field, indicate the number of In-Lines to be included in one set.

- In the **Y Start, Y end,** etc. fields, set similar parameters but this time for Cross-Lines.

- The **Bin Offsets** option activation switches on the offset summing mode.

- The offset start (in the **Off.start** field) and offset end (in the **Off.End** field) should be set.

- In the **Off.Step** the distance between the traces is defined in meters.

- In the **Off.Range** the interval around subsuming points is expressed in meters. All traces within this interval will take part in subsuming.

The Dataset... button opens the **Database Object Selection** dialog box, to, select a file from the database.

The **Save template** and **Load template** buttons save the module parameters in the form of templates into the project database and load the parameters from previously saved templates.

Data Input

This module has been designed to load external seismic data files into the flow. In most cases we recommend not to use this option. Instead, register a file in the project via the **Add data file...** command selected from the **Database** menu, and then load it into the flow via the *Trace Input* module. If you need to load an unregistered file into the flow and there is a specific module for the

file format, it would be preferable to apply the specific module rather than ***Data Input*** (for example, to load files in SEG-Y format it would be preferable to use ***SEG-Y Input*** instead *of* ***Data Input)***.

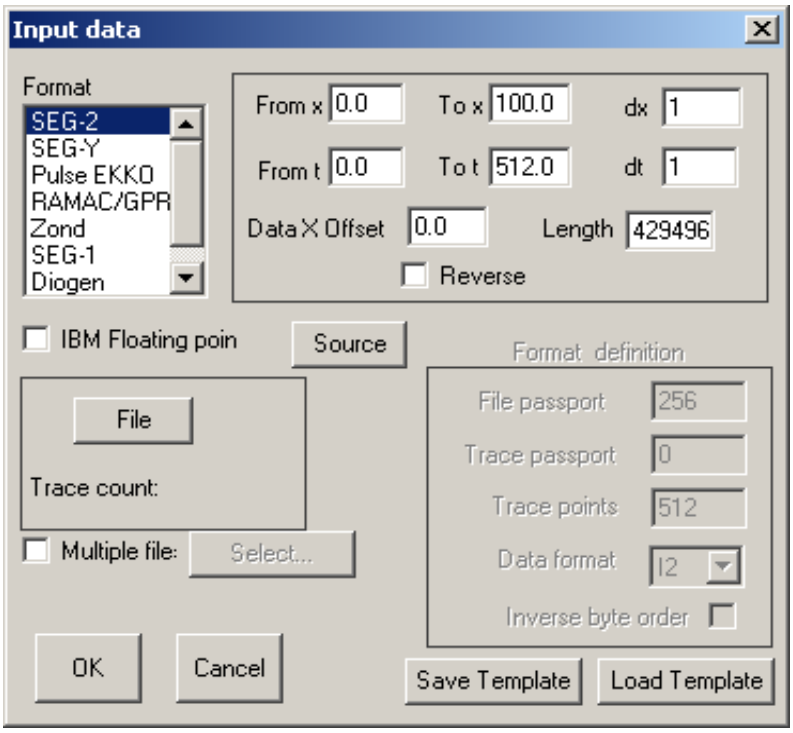
At present, the module supports 7 standard data formats (SEG-Y, SEG-2, SEG-1, Pulse EKKO, RAMAC/GPR, Zond, Diogen) and a user defined format. The user defined format support allows for reading of the data in any format having fixed file header and trace header lengths as well as fixed numbers presentation format , which can be one of the following: 8, 16, 32-byte signed integer, 32-byte real with floating point (in IEEE or IBM floating point standards). It is assumed that all numbers are recorded starting from lower byte to upper byte.

Information about sample interval (**dt**) and distance between traces (**dx**) is obtained automatically from the input file only when it is in one of the standard formats.

Parameters

The Data Input module has the following parameters:

- **Format** - select data file format out of seven possible formats (see above). By default the SEG-2 format is selected.



- **dx** - distance between adjacent traces expressed in meters

- **Length** - the profile length expressed in meters, varies according to **dx (Length=(n-1)*dx,** where n - is a number of traces in file)

- **dt** - sampling interval in time expressed in milliseconds

- **From t –** start time to be read from the file (0 - first sample in trace)

- **To t –** end time to be read from the file (if it is greater than the end time recorded in the file, then all data is read)

- **From x** - linear coordinate of the first trace to be read from the file (0 - first trace in file)

- **To x** - linear coordinate of the last trace be read

- **Data X Offset** - sets the offset (in meters) of the first trace of the data file with regard to x axis zero. In **From** x, **to** x fields the values should be set taking into account the **Data X Offset.** (For example, if **Data X Offset** equals 10, then the first trace in the file is associated with the X coordinate that equals 10 meters).

- **Reverse** - if the flag is switched on, then the data will be read from the file in reverse order.

- **IBM Floating point** - if the flag is switched on, then the numbers in R4 format are considered to be in R4 IBM 370 format, otherwise they are taken as R4 IEEE.

- **File** - by clicking this button you open a standard Windows dialog box for file selection. When the file is chosen the number of traces contained in a file will appear in the **Trace Count:** field. Here, the number of traces will be calculated according to the format that has been selected and, if the file format does not agree with currently selected format, the trace number may be defined incorrectly.

If one of the standard formats was chosen from the list then, after the file has been selected, the values obtained from file and trace headers will be automatically entered into *dx, dt, Data X Offset* fields (if the headers contain this information). After the fields have been automatically set, they can be changed by the user. In the flow the dx and dt values will be assumed as equal to those indicated in the ***Data Input*** parameters.

- **Format definition** - enabled when User Defined format is selected. It allows the user to define any arbitrary demultiplexed data format with fixed lengths of file and trace headers and fixed trace length. At that, only the data samples will be loaded from the file. The headers will be simply skipped and, therefore, all header information will be omitted. To define a format set the following:

- **File passport** - file header length (bytes)

- **Trace passport** - trace header length (bytes)

- **Trace points** - trace length (samples);

- **Data format** - number presentation format (I1, I2, I4 - 8, 16, 32-byte integers, respectively, R4-32-byte real with floating point).

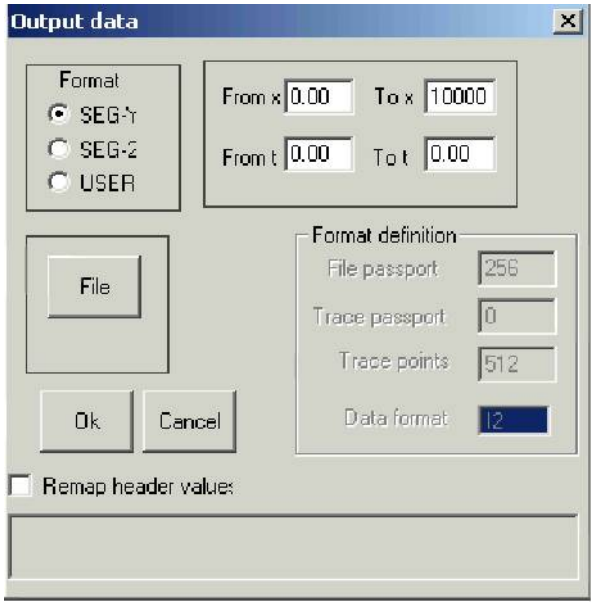
- **Save Template –** saves the dialog parameters as database template.

- **Load Template –** loads the dialog parameters from a database template.

Data output

This module saves the (processed) data from the flow into an external file on disk.

The dialog box for module parameters specification is similar to ***Data Input*** dialog box. Similar fields have similar meaning. The **Remap** option and corresponding text field designed for format remapping are similar to those of the **Add data file** (see **Add data file** chapter) tool.



In this case the **File** button opens a standard dialog box for selecting the name for the

file to which the data will be saved. This module saves the data in two standard formats (SEG-Y and SEG-2) and in the format defined by the user.

If a user defined format is used , apply format remapping to save **RadExPro** header information into the trace headers of the output file.

In SEG-2 format, the following extra fields will be recorded into the file header:

PROCESSING_DATE: file record date

PROCESSING_TIME: file record time

TRACE SORT AS_ACQUIRED:

UNITS METERS:

The following extra fields will be recorded into SEG-2 trace headers:

DELAY 0:

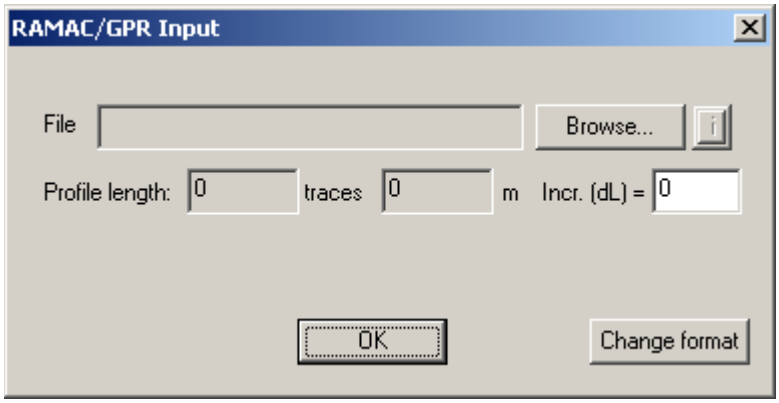
RECEIVER LOCATION: trace_number*dx

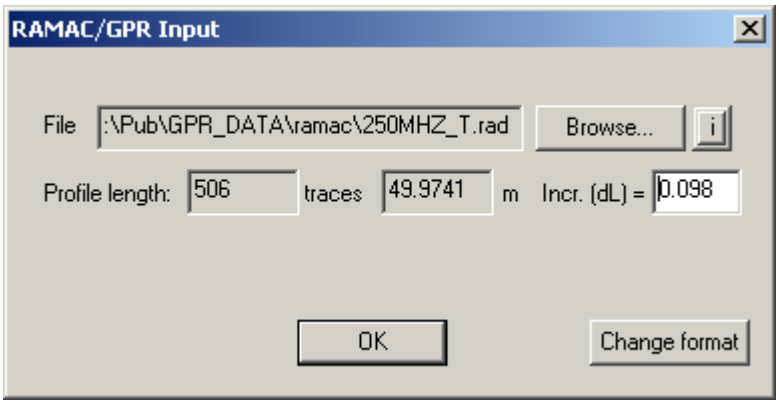
SAMPLE_INTERVAL: dt

WARNING: When saving processed data in a user defined format with integer types of data, pay attention to the dynamic range of the samples. For example, after automating gain control the sample values range from -1 to 1 and while saving this data in I2 number format the recorded samples will take on only three values - -1, 0, 1. To save this kind of data in integer formats you can use the ***Amplitude correction*** routine, for example, and set the Time variant scaling equal to a constant for the whole trace. The constant is defined from the ratio of flow data range to the range of integer format under use. (The I1 range is 127–128. The I2 range is –32,768–32,767. The I4 range is 2,147,483,648 – 2,147,483,647.)

RAMAC/GPR

When selecting the RAMAC/GPR format data input routine the following dialog box will appear.





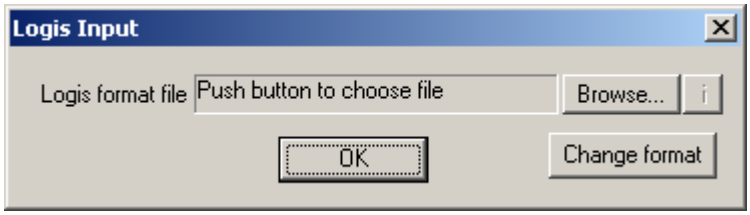
Click the "**Browse**" button to select a file in RAMAC/GPR (*.rad file extension). After that information about the file will appear in the dialog box fields.

In the "**Profile length**" group in the first field, the number of traces (GPR shots) recorded in the file will be displayed. In the second field, the profile length in meters will be shown, "**Incr. (dL)=**" will show the profile increment, i.e. distance in meters between the adjacent traces . In case if the profile increment is determined incorrectly you can change it by specifying the new value.

To confirm the data file selection click the "**OK**".

LOGIS

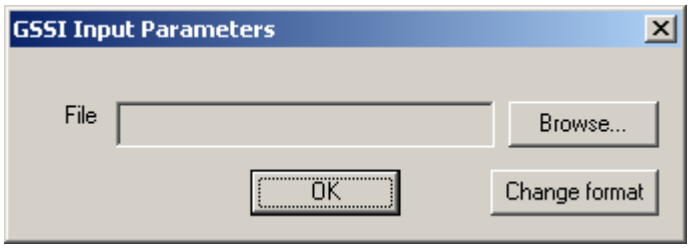
When selecting the GPR LOGIS (OKO) format data input routine the dialog box shown on the following figure will appear. Press the "**Browse**" button to select a GPR file in LOGIS format. If you click the "**i**" button the window containing information about the selected file will open.



To confirm the data file selection click "**OK**".

GSSI

When you chose the GSSI format of GPR data input the window shown on the figure below will open. Click the "**Browse**" button to select a GPR file in SIR GSSI (*.dzt file extension) format. The name of selected file will display in "**File**" field. To confirm the data file selection press "**OK**".

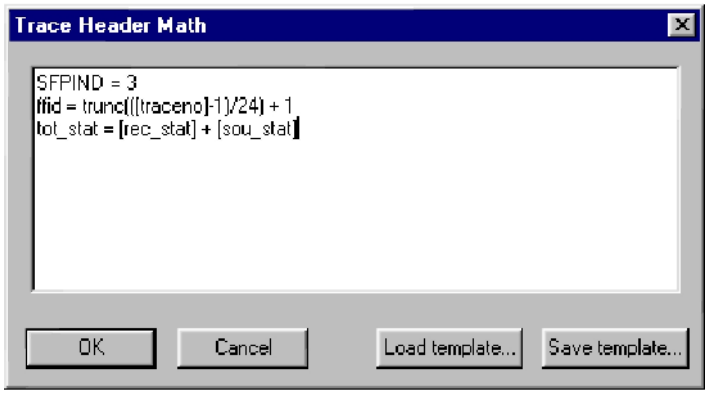


Geometry/Headers

Trace Header Math

The **Trace Header Math** module is meant for doing mathematical operations with values of existing headers. The operations should be specified in the form of equations.

When the module is activated the following window appears:



General syntaxes

<header name1>=<form1>

< header name2>=<form2>

A form may include digital constants, header field values, functions, mathematical operation:

- Digital constants:

Examples:

1 5.6 3.81e5

- Header fields' values

In the right-hand part of equations [<header name>] must be placed into square brackets *Examples:*

[sou_x], [source], [recno]

- **Mathematical operations:**

+ addition,

- subtraction,

* multiplication / division

^ involution

- **Functions:**

sin(x) - x sine;

cos(x) - x cosine;

tg(x) - x tangent;

ctg(x) - x cotangent;

arcsin(x) - x arcsine;

arcos(x) - x arccosine;

arctg(x) - x arctangent;

exp(x) - x exponent;

lg(x) - x common logarithm;

ln(x) - x natural logarithm;

sqr(x) -x^2;

sqrt(x) - x square root;

sh(x) - x hyperbolic sine;

ch(x) - x hyperbolic cosine;

sinc(x) = sin(x)/x;

abs(x) - x absolute value;

sign(x) - sign: gives -1, for x < 0, 0 for x = 0 and 1 for x > 0;

round(x) - gives an integer value closest to x;

trunc(x) - gives value of the closest integer, which is less than x;

ceil(x) - gives value of the closest integer, which is grater than x;

fmod(x, y) - residue of x division by y;

cond(c, x, y) - if c is true than gives x, otherwise gives y.

When specifying a condition you can use the following logical operations:

< - less,

<= - less or equal,

> - more,

>= - more or equal,

= - equal,

& - logical AND,

| - logical OR,

! - logical negation;

mindrv(x, y) - if x = 9999.0, it gives y, otherwise it gives x (equals cond(x=9999, y, x)) utmx(lat, lon, lon_m) / utmx(lat, lon, lon_m) - functions of geographical coordinates (lat - latitude, lon - longitude expressed in degrees, lon_m – longitude of the central meridian of the desired UTM zone) conversion into x/y coordinates - UTM meters.

Operation examples

sfpind=1

tot_stat = [rec_stat] + [sou_stat]

ffid = trunc(([traceno]-1)/24) + 1

chan = fmod([traceno]-1, 24) + 1

trc_type = cond(abs([disperse]) >= 0.8, 1, 2)

Save template and **Load template** buttons are meant for active equation saving in the template in the project database and for previously saved template loading respectively.

NOTE: The text buffer of the module can accommodate 200 characters. When a greater number of characters is typed, the buffer can become overfilled and an error message will appear. In this case,

reduce the number of operations. To fulfill the rest of operations, enter another copy of module into the flow.

Near-Surface Geometry Input

This module is used to assign geometry to field data obtained using the CMP, refraction seismic survey and surface wave analysis methods.

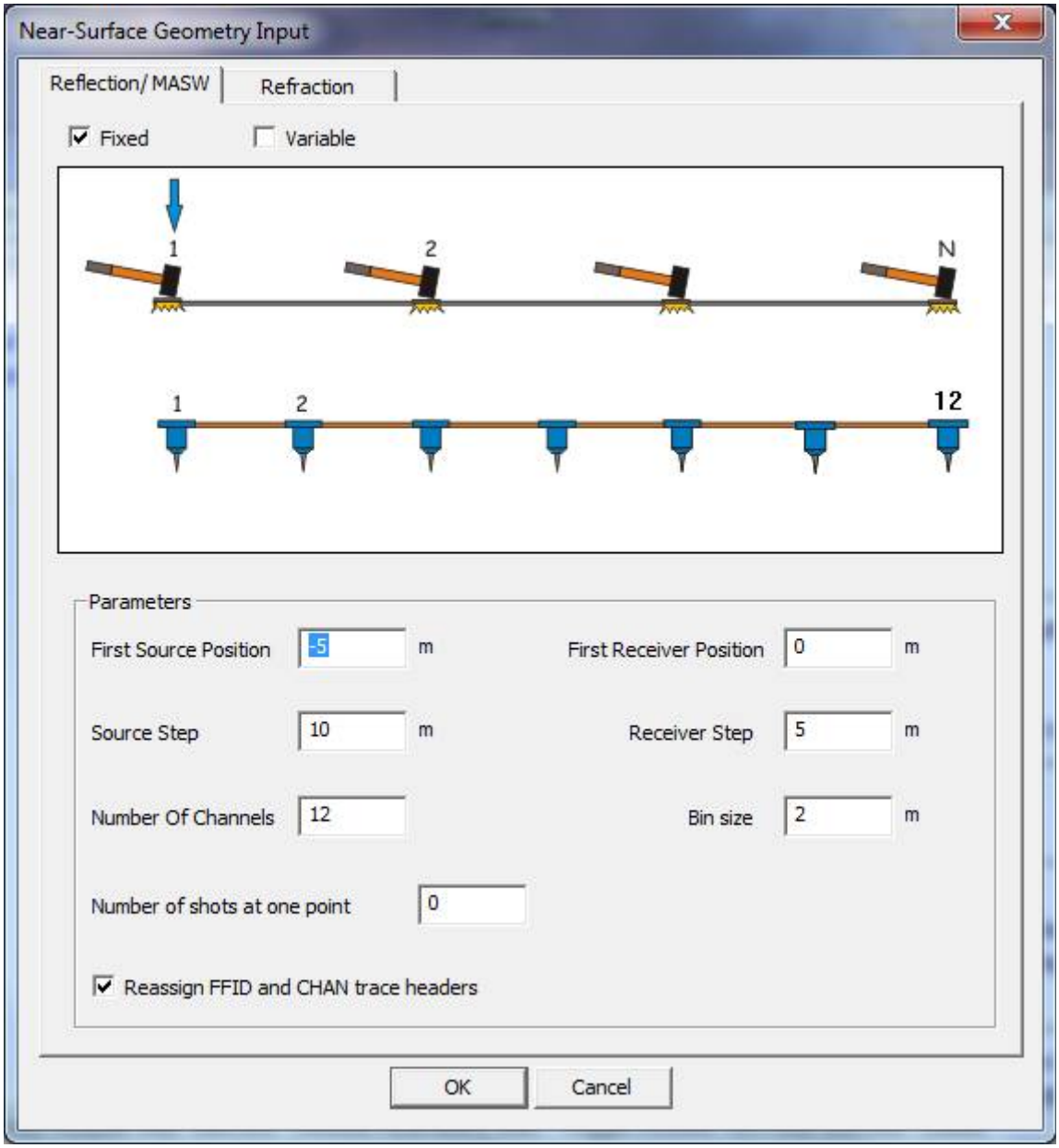
Module operation

For the module to function correctly, the data must be input in the order of their sequence along the profile, since the coordinates are calculated sequentially for each source position.

The following header fields are assigned to the source data set as a result of running the module:

- source coordinates – ***sou_x*** header field,
- receiver coordinates – ***rec_x*** header field,
- source to receiver distances – ***offset*** header field,
- CMP coordinates – ***cdp_x*** header field (calculated only if the **Reflection** tab is active),
- CMP numbers – **cdp** header field (calculated only if the **Reflection** tab is active),

The following procedure is recommended: load the data set using the **Trace Input** module → Assign the geometry using the **Geometry Input** module → Output the current data set into a separate data set with the assigned geometry using the **Trace Output** module.
The main dialog box of the module is divided into two tabs: assignment of geometry to data obtained using the CMP and surface wave analysis methods methods (Reflection/MASW) and assignment of geometry to data obtained using the refraction seismic survey method (Refraction). Each array type has its corresponding interactive image that visually displays the current parameter of the tab being edited (for example, receiver position, source position etc.). The module interface is shown below:



Module description

- **Reflection/MASW –** this tab allows assigning geometry to data obtained using the CMP and surface wave analysis methods.

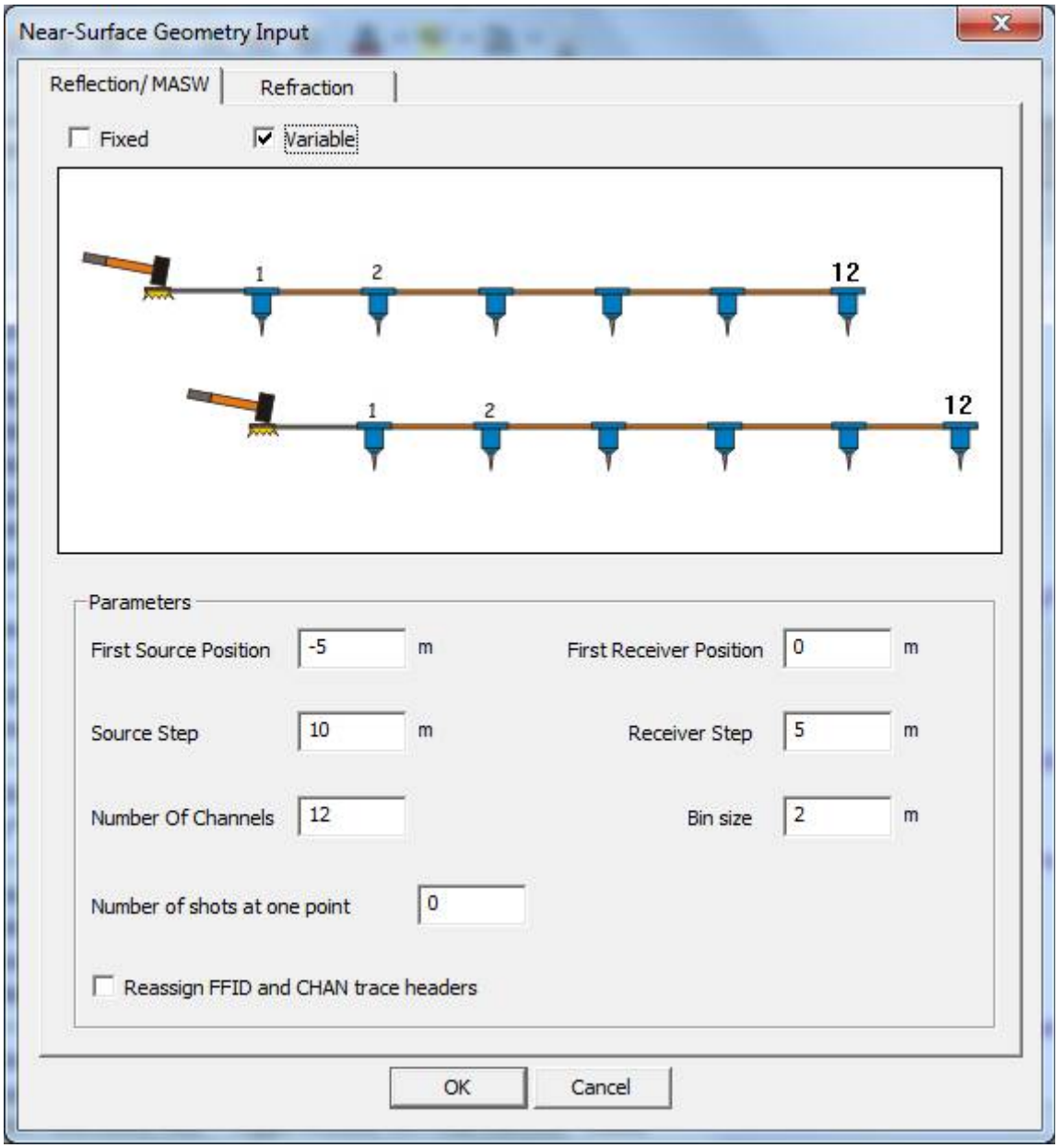
Fixed mode

This mode is used when the receiver array is fixed along the profile and the source positions are moved along the array. In this case the following initial parameters need to be entered in order to calculate the geometry (all distances are specified in meters):

- **First source position –** coordinate of the first source position.
- **Source step –** source position step.
- **Number of channels –** number of receiver array channels.
- **First receiver position –** coordinate of the first channel.

- **Receiver step –** channel step.
- **Bin size –** bin size. Data obtained using the CMP method need to have the CMP number specified in the *CDP* header field for further processing. Enter the preferred bin size in this field to calculate the corresponding header.
- **Number of shots at one point –** number of shots per one profile point. If several observations were made at a single profile point, specify their number in this field. This value may reflect both the number aggregations at the source position (if they were not stacked in the field) and different shot types (left/right). All shots at the same source position will be assigned the same source position coordinate.
- **Reassign FFID and CHAN headers –** if the header fields corresponding to the source position (FFID) and channel (CHAN) were left empty or were specified incorrectly, this option allows recalculating them based on the entered array parameters. Make sure that the data are input sequentially.

Variable mode



This mode is used to calculate the geometry of data obtained using an end-on array (the receiver array is moved together with the source along the profile at a certain step).

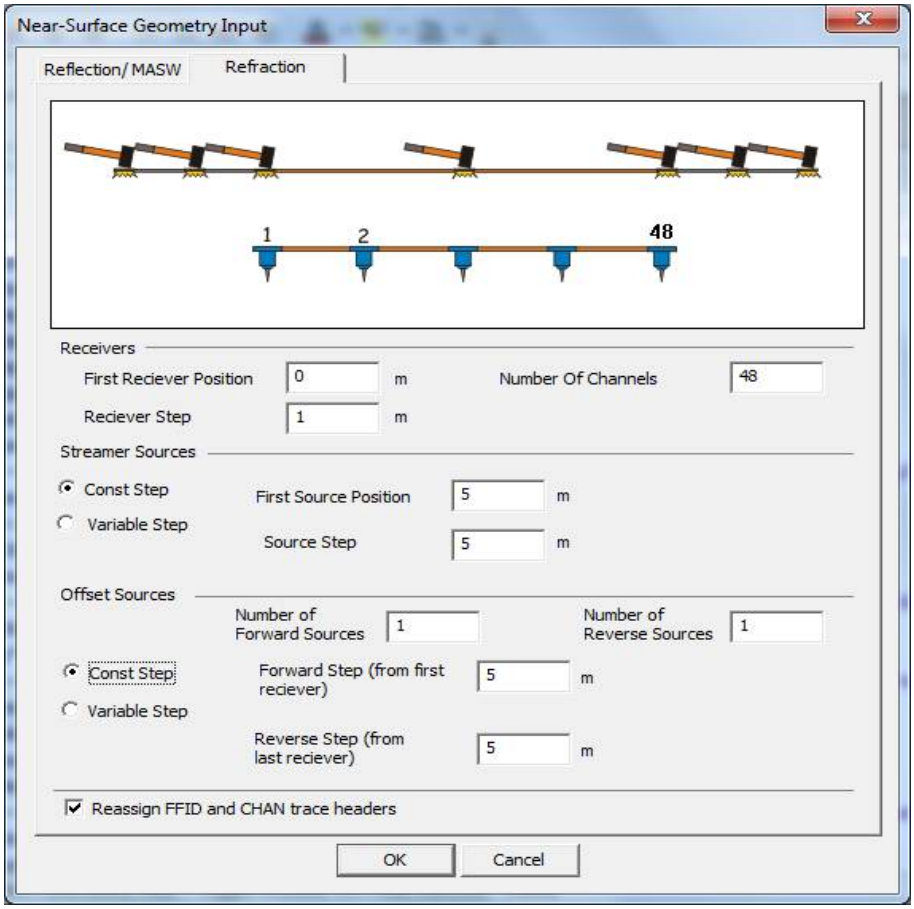
The following initial parameters are required to calculate the geometry:

- **First source position –** coordinate of the first source position.
- **Source step –** source position step.
- **Number of channels –** number of receiver array channels.
- **First receiver position –** coordinate of the first channel.
- **Receiver step –** channel step.
- **Number of shots at one point –** number of shots per one profile point. If several observations were made at a single profile point, specify their number in this field. This value may reflect both the number aggregations at the source position (if they were not stacked in the field) and different shot types (left/right). All shots at the same source position will be assigned the same source position coordinate.
- Reassign FFID and CHAN headers – if the header fields corresponding to the source position (FFID) and channel (CHAN) were left empty or were specified incorrectly, this option allows recalculating them based on the entered array parameters. Make sure that the data are input sequentially.

- **Refraction mode**

Refraction/MASW – this tab allows assigning geometry to data obtained using the refraction seismic survey method.

There are three main areas of the tab: 1) receiver line parameters, 2) offset source position coordinates , 3) streamer source position coordinates. The source position coordinates may be specified either with a fixed or with a variable step.



- **Receivers** – coordinates of the receiver array on the profile.
- **First receiver position** – coordinate of the first receiver.
- **Receiver step** – distance between the channels.
- **Number of channels** – number of channels in the receiver array.
- **Streamer sources –** coordinates of the source positions on the streamer.

If the source positions on the streamer were moved at a fixed step, use the ***Const Step*** option and specify the following parameters:

- **First source position –** coordinate of the first source.
- **Source step –** source position step.

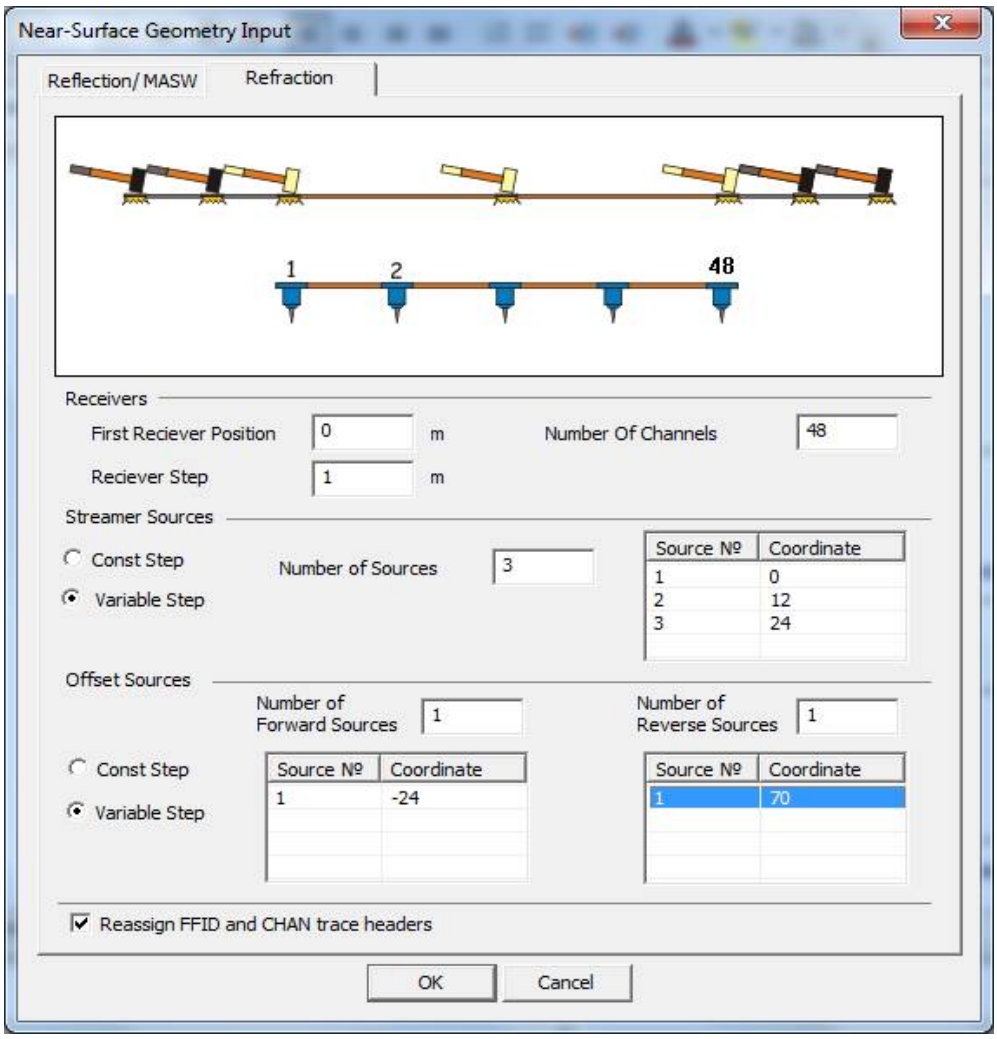
If the source position step was not fixed, select the ***Variable Step*** option. A table will open, allowing you to specify the source position coordinates on the streamer in the ***source position number – coordinate*** format. The number of lines in the table is the same as the value entered in the **Number of sources** field. Enter the coordinates of all source positions on the streamer (sequentially along the profile) into the table.

- **Offset sources –** coordinates of the source positions at offsets specified separately for the "forward" and "reverse" source positions. As in the previous case, the source positions can have either fixed or variable step.

To specify the coordinates with a fixed step, select the ***Const Step*** option and enter the following parameters:

- **Number of forward sources** – number of "forward" source positions.
- **Number of reverse sources** – number of "reverse" source positions.
- **Forward step** – "forward" source position step along the profile, starting with the first channel. For example: if the first channel coordinate is 0 and the number of source positions is 2 with the step of 5 m, the source position coordinates will have the values of -5 and -10 meters.
- **Reverse step** – "reverse" source position step along the profile, starting with the last channel.

To specify the coordinates of variable-step source positions, select the ***Variable Step*** option and enter the source position coordinates into the tables. The number of lines in the tables is the same as the values entered in the **Number of forward sources** and **Number of reverse sources** fields.



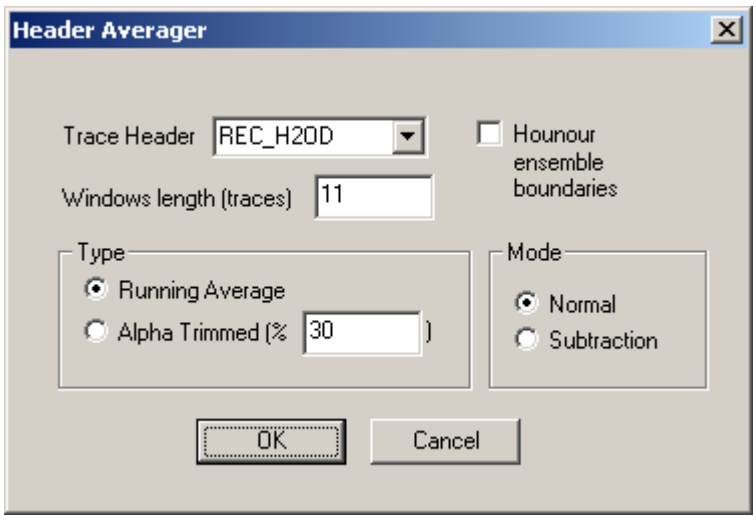
NOTE! Images displaying the various types of arrays are schematic and static. For example, the number of hammers representing the source positions does not change when the corresponding parameter (**Number of sources**) is changed on any of the tabs. Only a tip – a highlight or an arrow pointing to the parameter – is displayed when any of the parameters is changed.

Header Averager

The module is designed for averaging values of the indicated header in traces that appear in the flow. Averaging is performed in a sliding window of the given length. As a result of the module operation the raw header values are rewritten.

Parameters

Parameters' dialog is shown below:



Trace Header – the header, the values of which are going to be averaged

Window lenght (trace) – the length of the sliding window (number of traces), in which the averaging will be performed.

Honour ensemble boundaries – if this option is on, the averaging will be carried out within the ensemble boundaries independently.

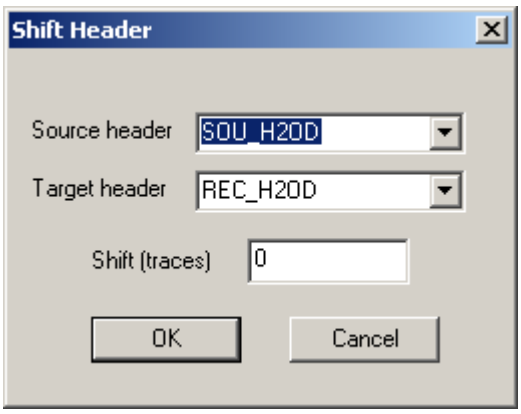
The group **Type** specifies the type of averaging: **Running Average** – running average, or **Alpha Trimmed** – trimmed average with the indication of trimming percentage.

Mode group specifies the mode of module operation: **Normal** – the value calculated for each position of the running window is written to the central point of the window, or **Subtraction** – the calculated value is subtracted from the value if the central point of the window.

Shift header

The module is designed for shifting values of the indicated header (**Source header**) in traces that appear in the flow by the indicated trace number **(Shift (traces)**). The shifted values are written to another header field (**Target Header**). The module can be used for example, in order to account for the distance between the GPS antenna and the seismic system while processing single channel off-shore data.

The parameters' window of the module is shown below.



The **Source header** value for the *i*-th trace in the flow will be written to the **Target header** of the trace **i-Shift** as a result of the module operation.

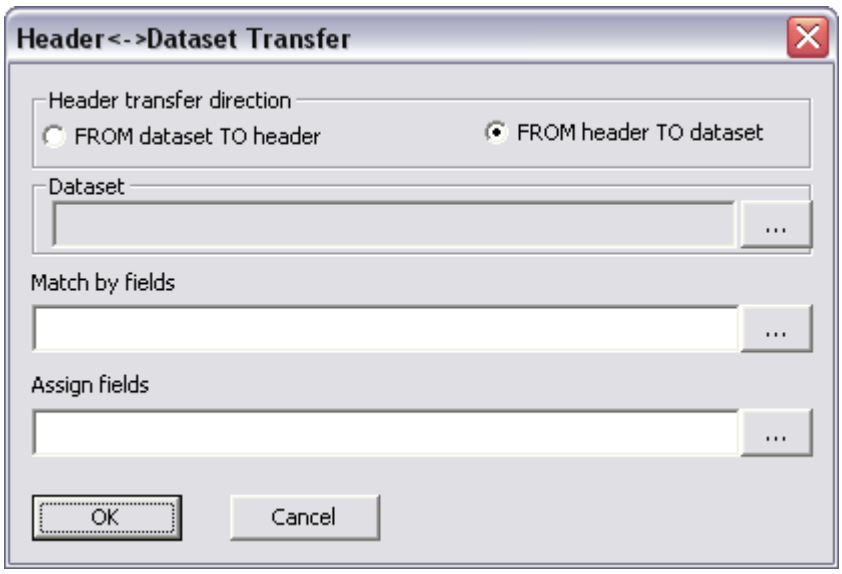
Header<->Dataset Transfer

The module is designed for exchange of trace header information between the data passing through the flow and a dataset in the project database.

In most cases, any changes made to the trace headers during flow execution are saved together with the seismic data to a new dataset by the **Trace Output** routine. However, this way of saving information in many cases is not convenient. When the only aim of a flow is to fill in/modify several trace header fields, without any processing of the seismic traces, it would be most logical to re-write the headers of the original input dataset rather than to create a new one with the modified headers. The **Header<->Dataset Transfer** routine provides necessary facilities for this operation.

The module can operate in 2 modes: (1) project header fields from the flow to a dataset, or (2) insert trace header values of a dataset to the header fields of the data in the flow.

The parameter dialog of the module looks as following:



Set up the mode of the operation in the **Header transfer direction** field: either **FROM dataset TO header** (header of the data in the flow) or **FROM header TO dataset**.

Click the **...** button to the right of the **Dataset** field to select a dataset to exchange trace header information with. The name of the selected dataset will be displayed in the string to the left of the button.

In the **Match by fields** field, select names of the trace header fields that will be used to match the traces from the flow to the dataset. You can type the header field names manually (separated by comma) or use the **...** button to select them from the list.

In the **Assign fields** field, select names of the trace header fields that will be assigned as a result of the module operation. You can type the header field names manually (separated by comma) or use the **...** button to select them from the list.

Example:

Assume, the module operates in the **FROM header TO dataset** mode, i.e. the values of the trace headers from the flow will be written to the dataset. The **Match be fields** string contains the following:

FFID,CHAN

The **Assign fields** string looks as following:

PICK1,PICK2

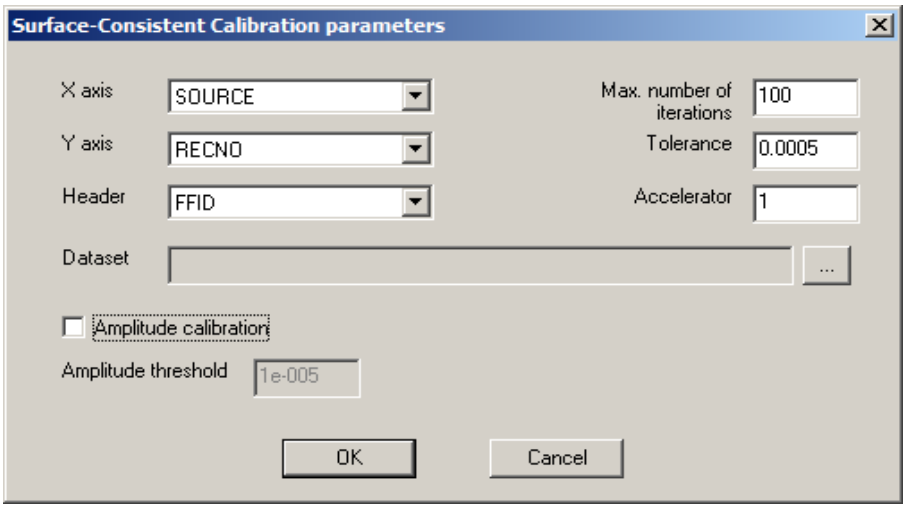
As a result of the module operation, for each trace in the flow the following operations will be made:

- In the dataset, all traces with FFID and CHAN equal to those of the current trace in the flow will be found.

- For all these traces in the dataset, PICK1 and PICK2 header fields will be assigned by the values of the PICK1 and PICK2 header fields of the current trace in the flow.

Surface-Consistent Calibration*

Surface-Consistent Calibration is a stand-alone module that allows surface-consistent calibration of the values stored in a trace header field (static shifts or amplitudes). The calibration is performed within limits of bifactorial model. In this model it is assumed that distortion of a value (static shift or amplitude logarithm) is a superposition of 2 independent distortions related to 2 axes – X and Y (typically, those would be distortions related to shooting conditions and distortions related to receiving conditions).



In the **X axis** field specify a header word indicating a source of the first distortion (e.g. source point).

In the **Y axis** field specify a header word, indicating a source of the second distortion (e.g. receiver

point).

In the **Header** field specify a header word containing values that are to be calibrated.

Click **...** button to the right of the **Dataset** field to select a dataset which header values are to be calibrated.

Amplitude calibration – when selected this switch indicates that the calibrated values are to be considered as amplitudes. In this case, the logarithm of the original values is calibrated.

Amplitude threshold – the values lower than the threshold are not used for calibration.

If it is need to make the calibrating process iterative, **Max number of iterations, Tolerance** and **Accelerator** parameters are to be used. Note that the duration of the iterative calibration process increases proportionally to square of the number of seismic traces. For acceleration of this process one could slightly decrease the number of conditions under test within each iteration increasing the **Accelerator** parameter (1 = no acceleration).

The breaking iterations condition could set through the **Tolerance** parameter or indicated directly as a **Max number of iterations**.

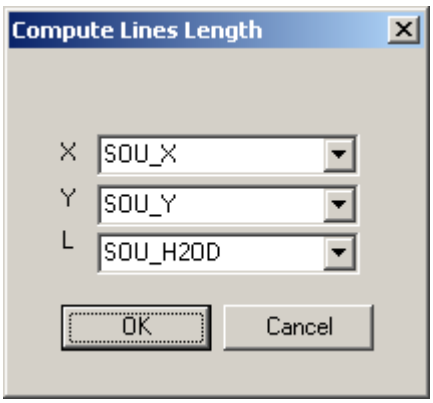
Compute Line Length

The module is dedicated to calculation of the cumulative distance of each trace from the beginning of the line. For the i-th trace in the flow, its distance L(i) is evaluated as:

$$L(i) = L(i-1) + dL_{(i-1,\ i)}.$$

where $dL_{(i-1,\ i)}$ – distance between trace *i-1* and trace *i*.

The parameter dialog looks as following:



You shall indicate the header fields with the **X** and **Y** coordinates of each trace are stored, which will be the input for distance calculations, as well as the output header field where the calculated **L** values will be stored.

Interactive Tools

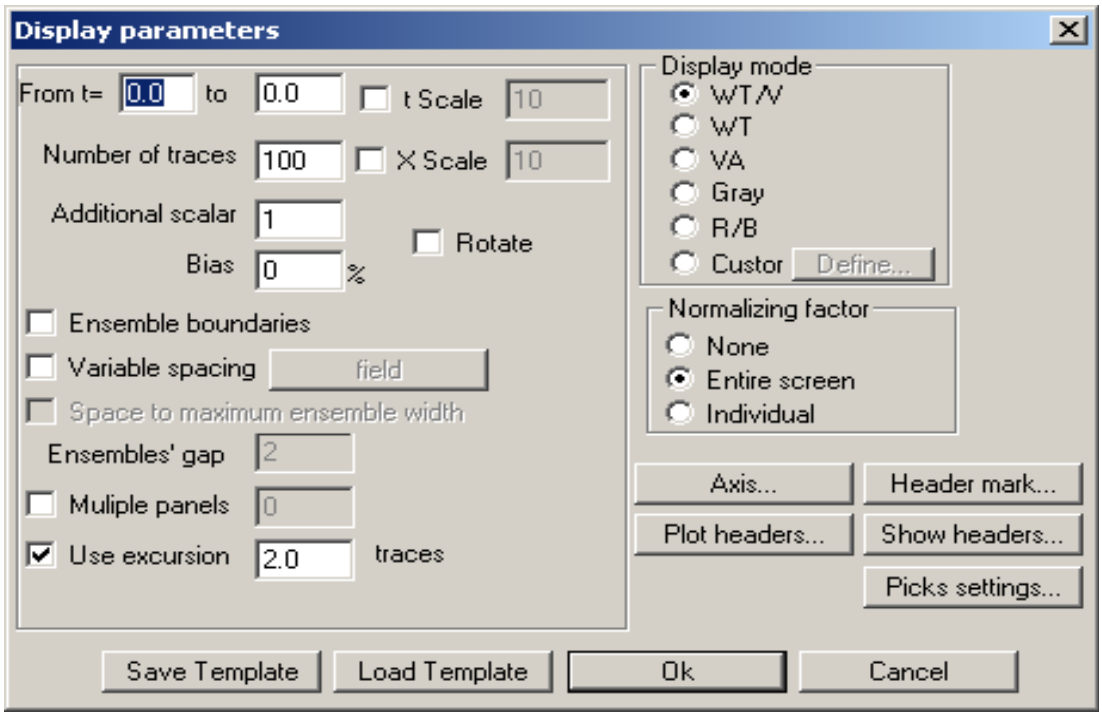
Screen Display

The **Screen Display** module is an interactive application used for visualization of data through-passing the flow in the form of time sections. It is also possible to view the data on several panels, placed one under another. When doing this, the images on the panels can be superposed according to the position of receivers, sources, CDP points. The application has a number of features that allow the user to carry out various operations with headers, interactive static correction, reflection picking, data import/export in the form of text files, etc.

The interface of the module consists of two parts: the dialog for initial parameters setting and the main running window opened by the module during its execution. The dialog box for parameters specification can also be opened while operating the module, but in this case all the parameter changes will be active only during the current module execution session.

Parameters

The **Display Parameters** dialog box for the **Screen Display** parameters setting looks like the following:



- **From t=:** The start time expressed in ms. The time sections will be displayed on the screen starting from this time (all data can be viewed by scrolling).

- **to:** The end time expressed in ms. The time sections will be displayed on the screen up to this time. To display all the samples to the end of the trace, enter 0. This parameter is active if the **tScale** option is switched off. (All data can be viewed by scrolling).

- **Number of traces:** Number of traces displayed on the screen. This parameter is active if the **XScale** option is switched off.

- **Tscale:** If this option is switched on, the data will be displayed in a vertical scale specified by the user in explicit form. In a corresponding field specify a vertical scale value expressed in ms/cm.

- **Xscale:** If this option is switched on, the data will be displayed in a horizontal scale specified by the user in explicit form. In a corresponding field specify a horizontal scale value expressed in traces/cm.

- **Additional scalar:** This is an additional scalar (screen gain factor) on which the trace sample values will be multiplied before being displayed on the screen.

- **Bias:** In this field, specify a value of the average trace level bias from zero. In case of variable width mode the parameter changing results in a change in the black color level. The positive value will cause displacement to the left from the zero line of the trace with enlargement of the blackened area of the curve. The negative value corresponds to a reduction of the blackened area of the curve. When **Gray**, **R/B**, or **Custom** display mode is selected, this value will shift the zero level of the data relative to the center of the color palette.

- **Ensemble boundaries:** You should activate this option in order to make the *Screen **Display*** divide different ensembles by gaps. The ensemble key is the first sort-key specified in the module of data input into the flow.

- **Variable spacing:** Select this option to space the traces on the screen in accordance with specified header values. For example, you can use this option to display stack traces according to their real coordinate along the line. To select the header, click the **field** button.

- **Space to maximum ensemble width:** If this option is activated and the parameter **Multiple panels** > 1, the traces in every panel will be spaced within the headers range of that ensemble which has the maximum number of traces. This clause is important in case you do analysis of the sections in several panels placed one under another and it is required to space the traces in every panel in the same order dictated by selected header value of, for example, the receiver. When this option is activated the traces from various sections with the same receiver number will appear directly one under another in different panels. This option is available if the **Variable spacing** option is switched on.

- **Ensemble's gap:** The width of the gap between traces ensembles. This field is available if the **Ensemble boundaries** option is switched on.

- **Multiple panels:** You should switch this option on in order to make it possible to work in multiple panels mode. The number of panels should be entered in a corresponding field.

- **Use excursion ___ traces:** This option is used when the data is visualized in WT, VA or WT/VA modes for limitation of maximum number of traces to be crossed by the current trace before its amplitude is clipped. Specify the maximum acceptable number of traces to be crossed. For example, if this option is toggled on, and the number of traces is 2, the amplitudes of the current trace (either

negative or positive) will be clipped when crossing the second trace away from the current trace.

- **Display mode:** Select the trace display mode. These are the possible variants:

- **WT/VA** displays the traces in wiggle trace /variable area mode

- **WT** displays the traces in wiggle trace mode

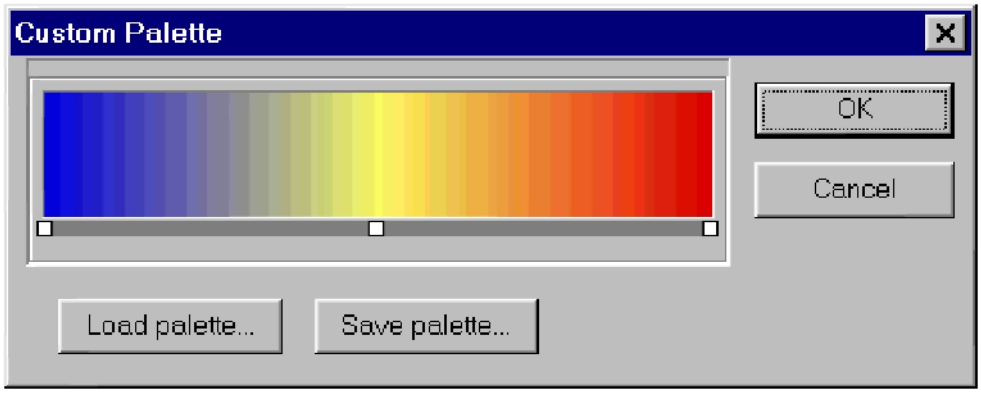
- **VA** displays the traces in variable area mode

- **Gray** displays the traces in variable density mode in grayscale

- **R/B** displays the traces in variable density mode in red-white-blue palette

- **Custom** displays the trace in variable density mode in the palette defined by the user

- **Define** button is available only when the **Custom** option is switched on. The **Custom Palette** dialog box will open:



The pallet should be defined as a set of points with specified colors. Colors are linearly interpolated between the set points. You can create, replace, or delete points and change a points' color. The points are marked under palette image on the gray bar (white rectangles).

- **To change a point position,** click on it by the left mouse button (**MB1**) and, holding it, drag to the required position and release the button. The palette image will be changing while the point is moved.

- **To change the color of an existing point,** double-click on it by the left mouse button (**MB1**).

A standard dialog box for point color changing will open.

- **To create a new point with specified color,** click on the required place by the left mouse while holding down the Shift key (Shift+**MB1**). A standard dialog box for point color selection will open.

- **To delete a point from the palette,** click the right mouse button (**MB2**) on it.

- **Load palette...** button opens a standard dialog box for opening a disk file containing palette definition,

- **Save palette** ... button opens a standard file saving dialog for saving the active palette to a file on disk.

The palettes are stored on disk in ASCII files, defining the position of the palette points and their RGB colors.

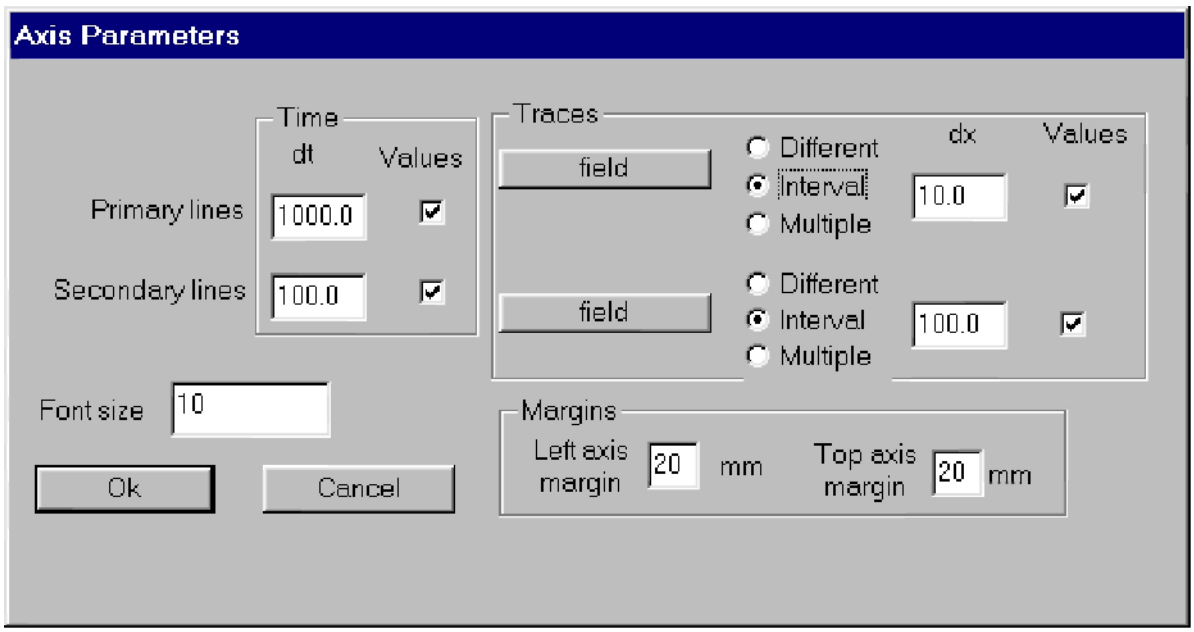
- **Normalizing factor:** Select amplitude normalizing mode for traces to be displayed on the screen. The following variants are possible:

- **None:** there is no additional trace normalization,

- **Entire screen:** all traces are normalized in the aggregate by means of dividing the trace amplitudes by mean absolute value of amplitude of all traces,

- **Individual:** normalizes all traces individually by means of dividing the trace amplitudes by mean absolute value of amplitude of the trace itself.

- **Axis** button opens the **Axis Parameters** dialog box for axes parameters specification:



The dialog box includes the following fields:

Time. In this field, specify the time intervals in which primary (thick) and secondary (thin) lines of time grid will traverse. For example, to display the thick lines in every 1000 ms and thin lines in every 100 ms, enter 1000.0 in the **Primary lines** field and 100.0 in the **Secondary lines** field.

Values: If this option is activated, the lines of corresponding type will be labeled on a vertical scale to the left of the time section.

Traces: This group of fields sets the horizontal scale label's parameters—trace labels. To select a header field, the values of which will be used as the trace label, click the **field** button. It is possible to use two trace labels. In this case every label should be adjusted independently. You can specify the trace label's frequency, i.e. the interval of labels displaying on the horizontal scale. The following variants are possible:

Different: labels the first trace and then every trace for which the header value differs from that of the previous trace .

Interval: labels the first trace and then every N-trace after it. Here, specify the desired interval N in the **dx** field. For example, if you enter 2 in the **dx** then the traces will be labeled as 1, 3, 5, 7, etc.

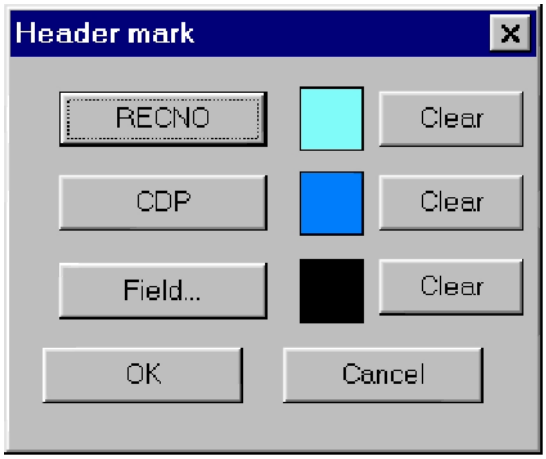
Multiple: labels the trace if the header falls on a number divisible by the selected increment, which should be specified in the **dx** field. For example, if you enter 5 in the **dx** field, then the labels will be on all traces for which the header value is 5, 10, 15, 20, etc.

Values: To display the trace labels on the screen, activate this option.

The font size to be used for label displaying on the screen can be set in the **Font size** field.

Margins: Here you can adjust the dimensions of the fields to the left and above the time section image on the screen. These fields will be used for trace axes and labels. Field dimensions should be expressed in mm.

Header mark...: By clicking this button the **Header mark** dialog box for header marks parameters specification will open:



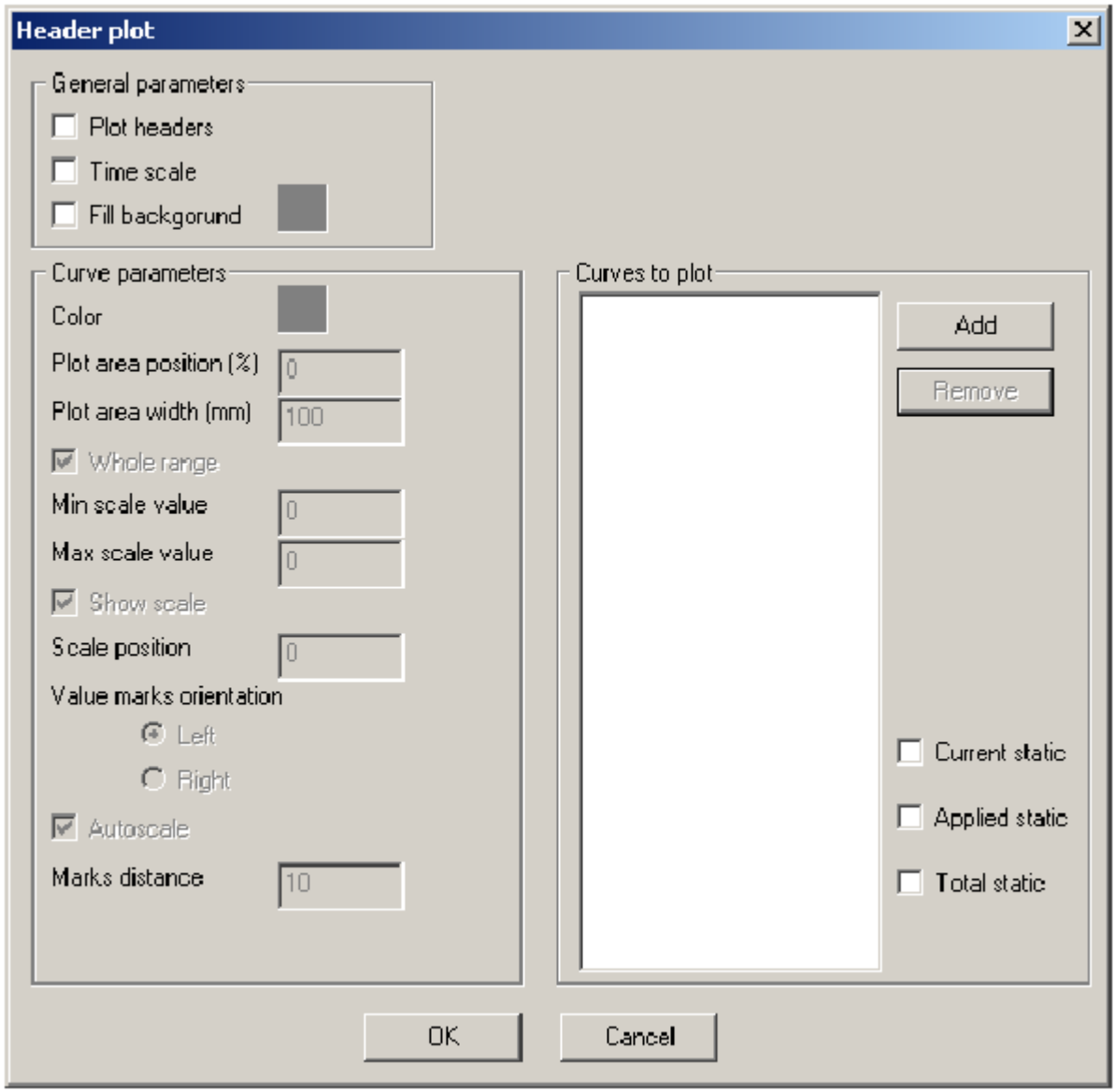
Field....: click this button to enable selection of the header field to which the present type of marks will refer. You can use marks which refer to three headers at the same time.

Select the header word from the list and then click on the square to the right for mark color selection. A vertical line of the specified color will mark all traces where the value of the selected header changes.

Clear: This button removes the selected header word from the **Field...** and cancels all marks that refer to it.

Plot headers...: When clicking this button the **Header plot** dialog box meant for setting the

parameters for selected header field values charts displaying on the screen will open:



- **General parameters:** This is the group of general parameters, common for all curves.

- **Plot header**s: Toggle on this option, if you wish that the curves of the header values are visualized on the screen of the module **Screen Display**.

- **Time scale:** Toggle on this option, to regard the header values as time values and put them on a time scale of the seismic data. For example, you may plot the values of static shift on real data scale using this option.

- **Fill background:** This option is used for plotting the curves on a background of solid fill. Click on a colored square to the right to choose the background color.

- **Curves to plot:** You may indicate here which headers should be plotted as curves.

- **Add:** Use this button to choose the header fields to plot. A list of headers available will appear when you click the button. You may choose several headers from the list using the left mouse button and the buttons Shift and Ctrl. After selecting the headers, click OK. The selected headers will appear on the list.

- **Remove:** This button removes the selected header fields from the list of plotted curves. You may select several headers simultaneously from the list using the left mouse button and the Shift and Ctrl buttons. Moreover you can plot interactive static corrections as graphs

- **Current statics.** Toggle on this option, if you want to plot on the screen the time shifts made interactively in the current panel, but not applied yet.

- **Applied statics**. Select this option, if you want to plot on the screen the time shifts applied to the data in the panel. Those shifts do not influence the **Current Statics**.

- **Total statics.** Toggle on this option, if you wish to plot the total static shifts, i.e. the sum of the **Current Statics** and the **Applied Statics**.

- **Curve parameters:** The individual parameters of plotting each of the header fields from the list are adjusted here. Click the left mouse button on one of the selected header fields from the list to view and edit the plot parameters.

- **Color:** The color of the selected header field plot is set here. Click on the colored square to set a new color. The scale of curve representation is assigned in the underlying fields. They are accessible, if the option **Time scale** is toggled off.

- **Plot area position (%)**: Enter the value of header curve downward displacement in percentage from the screen size. If the field is set to 0, the curve will be plotted directly above the time section.

- **Plot area width (mm):** Plot area width (in mm), to which the graph of the selected header field will be output.

- **Whole range:** When this option is toggled on the whole range of the header value is adjusted to the selected plot area. Otherwise you should set the range of plotted values manually, selecting the minimum (**Min scale value**) and the maximum values (**Max scale value**).

- **Show scale:** This option allows plotting the axis with the scale for a selected graph. If it is toggled on, you may set the axis position in percents from the screen in field **Scale Position** (0 – from the left, 100 – from the right).

- **Value marks orientation:** allows setting the position of captions, relatively to the scale axis - to the left (**Left**) or to the right (**Right**).

- **Autoscale:** When this option is toggled on, the interval between the scale marks on the axis is selected automatically. Otherwise, you have to set the needed interval in the field **Mark distance**.

After adjusting the plot parameters of one of the header fields from the list, you may switch to the next field from the list, using left click. The parameters adjustment for the previous field will be saved. After setting the parameters, click **OK**. The values of the selected header fields will be plotted as graphs, in compliance with the chosen parameters.

- **Show headers...:** When clicking this button the **Quick show header fields** dialog box for selection of header fields, for which values will be displayed in the status bar, will open:

- **Add...:** click this button to choose the header, for which values should be displayed in the status bar, from the header fields list. The header fields selected to be displayed will be included in a list in the box on the left side of the dialog window.

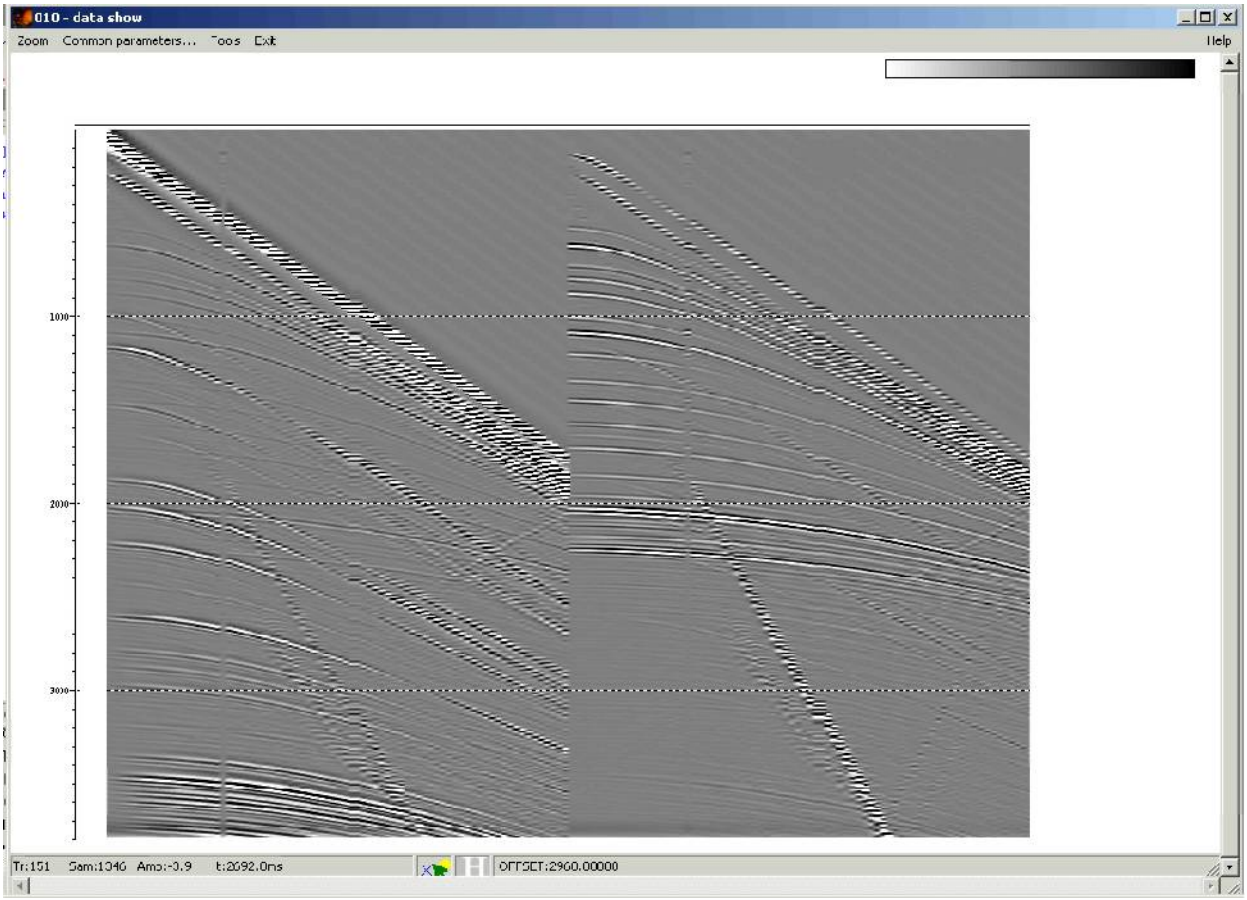
- **Delete:** This button allows deleting the header from the list of headers selected to be displayed.

- **Quick show dataset name.** If this option is switched on, the name of the currently displayed dataset at the mouse cursor position will be shown in the status bar.

When parameters are specified, click the **OK** button. The values of the selected header fields of the trace on which the cursor is placed will be displayed in the right part of the status bar of the **Screen Display** module running window.

Working with Screen Display module

When starting execution of the flow containing **Screen Display** module the window, similar to the one shown on the figure, will appear:



Menu

The module menu contains the following submenus:

Zoom

Display parameters

Tools

Exit

Help

In addition, when clicking the right mouse button (**MB2**) on any place of the screen a pop-up menu similar to the **Tools** menu will open.

- **Zoom:** The menu contains commands for zooming in the interactively selected data image fragment to the sizes of running window.

Set zoom: In order to zoom in a data fragment, select this command (or press **Z** hot-key) and then click the left mouse button (**MB1**) on a corner of the fragment to be enlarged and, holding the button down, move the cursor to another corner of the fragment and then release the button. The selected rectangle-like data fragment will be enlarged to the size of the whole window.

Unzoom: (**X** hot key) Returns the display mode to initially set scale.

Local zoom: This command is used to enlarge a data fragment in a separate window. The way to use this command is similar to that of the Set zoom command. Via this command you can open several enlarged fragments in separate windows.

- **Common parameters:** This submenu changes the *Screen Display* module parameters specified during flow creation. When selecting this command the dialog window **Display Parameters** (see the **Parameters** section) will open.

Working with the flow you can change the data display mode without calling **Display Parameters** dialaog, but using instead the following hot-keys:

Ctrl+1 – WT/VA mode

Ctrl+2 – WT mode

Ctrl+3 – VA mode

Ctrl+4 – Grey mode

Ctrl+5 – R/B mode

Ctrl+6 - Custom mode

- **Tools:** This menu contains the following commands:

Approximate: contains the tools for estimation traveltime parameters(velocities) of different kinds of waves, by approximation of their travel-time curves.

Hyperbola (diffraction): allows estimating the effective velocity and the depth of a diffracting object from traveltime curve of diffracted wave on stacked seismic section. When you select the command a hyperbola – theoretical traveltime curve of a diffracted wave is plotted on the screen. The traces' coordinates, recorded in the fields CDP_X and CDP_Y are used during the theoretical traveltime curve calculation. To determine traveltime parameters of an observed diffracted wave you have to place against it a theoretic "hyperbola" and insure their maximum fit. In order to manage the theoretic "hyperbola", use left and right mouse buttons. A text line with current values of effective velocity and depth to diffracted object will be displayed in the upper part of the screen.

Hyperbola (reflection): is intended for estimation of reflected waves' effective velocities. This tool is to be applied to unstacked data. When you select the command a dialog window appears, in which you have to indicate the hyperbola parameters: steps and ranges for the vertical time (**t0 step**, **min**, **max**), effective velocity (**v step**, **min**, **max**) and dip of reflecting boundary (**fi step**, **min**, **max**). The dip angle is given in degrees; positive values correspond to the boundary dip towards offset increase. Once the parameters are set, click **OK**. The theoretical traveltime curve of reflected wave will be plotted on the screen. The calculation process of the traveltime curve uses offsets (in meters), which are supposed to be stored in the header field OFFSET. To determine the traveltime parameters of an observed reflected wave you need to achieve a maximum match of an observed traveltime curve with theoretic hyperbola. A text line with current parameters of the theoretic traveltime curve will be reflected in the upper part of the screen. In order to change the parameters of the "hyperbola" use the buttons:

left/right arrows – change the effective velocity

up/down arrows – change vertical time

+/- - change dip angle

Line header word...: This command is used for **Line** tool adjustment, intended for apparent velocity estimation. The header fields' selection dialog opens when you select the command. You have to choose the header field, containing the distances, which will be used for calculation of the apparent velocities. For example, for CDP data processing it could be offset, written to the OFFSET

field, for VSP data processing, it could be a field, containing cable depth values.

- **Line:** This tool is intended for the estimation of apparent velocities, as distance increment to time increment ratio. The values are taken from the header that had been chosen beforehand via the **Line header word...**. command.

- **Save parameters:** If you use the command when one of the traveltime estimation tools is active, a window with an editable text field will appear on the screen. Press Ctrl+Q to copy all current parameters of the active tool as a line to the text window.

- **Spectrum**

- **Average:** This command allows you to display an averaged spectrum of a group of traces within a specified window. The window is set using the same mode as the **Zoom** mode (see above). The average spectrum of the selected portion of the seismic data appears in a separate window. To fit a sector of the spectrum to the size of the whole window click and hold the right mouse button **(MB2)** at the beginning of the spectrum and drag the cursor to the end of the sector. To return to the initial spectrum scale, click the left mouse button **(MB1)** twice in any place of the spectrum window. A pop-up prompt with the frequency value (in Hz) and amplitude corresponding to this frequency in percents will appear when you click the left mouse button **(MB1)** in the spectrum window. Several spectrum windows can be viewed simultaneously. To remove the spectrum window, close the window. If there is at least one spectrum window on the screen, an additional command **Print spectrum...** appears in the menu . This command is used to print the active spectrum window.

- **2D Spectrum:** This command displays F-K spectrum of group of traces within a specified window. The window is chosen using the same method, as you used to set **Zoom** (see above). 2D spectrum is viewed in colors, corresponding to the current palette, set for traces visualization. If the traces are visualized using the wiggle traces method or/and method of variable area, a gray palette is used. **Additional scalar** parameter, set in the **Display Parameters** affects the spectum visualization as well. Wave numbers **k** (in mm^{-1}) are depicted on the horizontal axes, while frequencies in Hz/MHz are depicted on the vertical axis of the spectrum window. The line for dips estimation passes from the point of origin. The dip of the line can be changed using the left mouse button **(MB1)** click within the spectrum visualization area. The numerical value of current dip is shown in the left corner of the spectrum window.

Pick — This is a **Tools** menu item that contains the commands for manipulations with picks.

New pick: This command can also be launched by pushing the N key on the keyboard. It allows switching to horizon picking mode and creating a new pick. The rectangular icon in the status bar will display the color of the new pick. By repeating the command you can set as many picks as needed.

The commands of the picking mode:

•A pick node can be **added** by clicking the left mouse button (**MB1**). The node appears at the place of cursor position.

•A pick node can be **moved** to a new place using the right mouse button (**MB2**). You can either drag the node to a new location keeping the right mouse button pressed, or just right-click on the desired new location – in this case, a node closest to the cursor position will be moved to the specified location.

•A pick node can be **deleted** by double-clicking the right mouse button (**MB2**). By doing this the node closest to the cursor position will be deleted.

•**Active pick changing (selection)** can be done in three ways: 1) by pressing the Shift key simultaneously with the left mouse button (MB1) at any node of the required pick, or 2) by successive pressing of the **Tab** key till the required pick becomes active or 3) by successive selection of the **"Next pick"** and the **"Previous pick"** commands in the **"Picks"** menu.

•**Exiting the mode/entering the mode** can be accomplished in three ways: 1) by pressing the N key on the keyboard (entering the mode simultaneously with creating a new pick), or 2) by selecting the **Pick mouse mode** command from the **Tools/Pick** menu, or 3) by clicking the rectangular pick icon on the status bar.

Delete pick: This command can be activated either through the **Tools** menu or by pressing the **Del** key on the keyboard. This command deletes the active pick entirely or in fragments. To delete a pick fragment, place the cursor above the upper horizontal axis at the position corresponding to one of the selected data area boundaries and then click the left mouse button

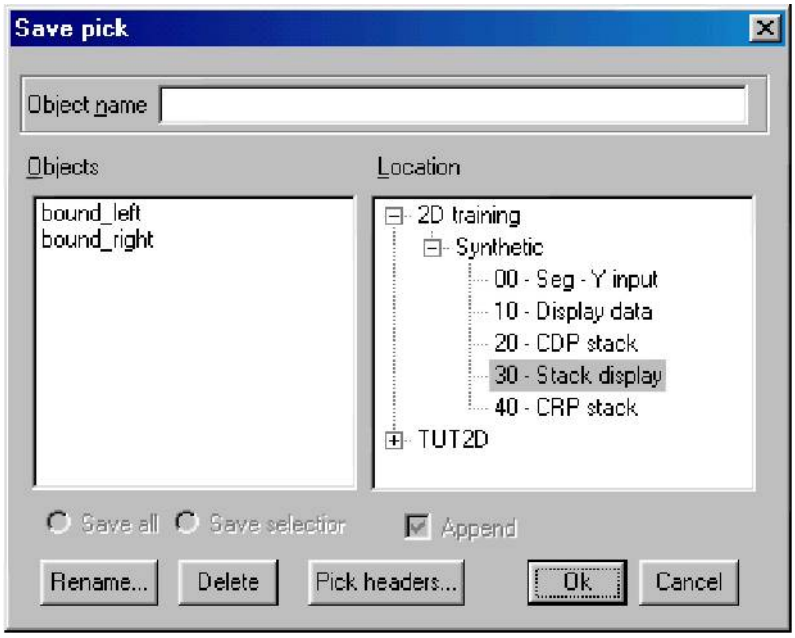
(**MB1**) and, holding the button pressed, move the cursor to another boundary. When releasing the mouse button the selected area will be marked by color. (To undo the selection, click the left mouse button (MB1) above the horizontal axis). Next, when selecting the **Delete pick** command, all points of the pick within the selected area will be deleted. If a pick fragment has not been selected the entire active pick will be deleted.

- **Load pick:** This command can be activated either from the **Tools** menu or by pressing the **Ins** key on the keyboard. This command allows for loading a previously saved pick .

- **Load pick with interpolation:** This command can be activated only from the **Tools** menu. It allows loading the pick from the database with linear interpolation between the original pick points.

- **Save:** This command can be activated only from the **Tools** menu. It allows saving the pick into the database under its existing name. If this pick is not in the database yet, then this command can be activated similarly to the **Save as** command.

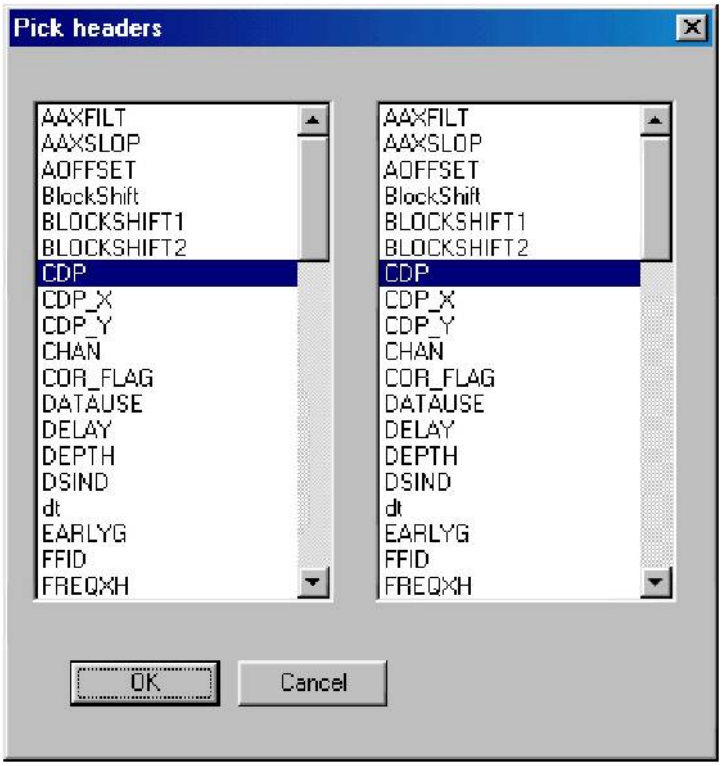
- **Save as:** This command can be activated either from the **Tools** menu or by pressing CTRL-C keys combination. It allows saving the pick into the database under new name. When selecting this command the dialog box for database object selection appears:



However, in addition to the standard fields of the **Database Object Selection**

dialog box there are several additional features in the **Save pick** window.

In order to save the pick into the database, specify two header fields to which the picked time values will be bound. To do this, in the **Save pick** window, click the **Pick headers button** and, in the opened window with two headers lists, choose the required fields:



Switching on the **Save all** option in the **Save pick** window enables saving of the whole pick, while the **Save selection** option saves the selected pick fragment only. The **Append** option allows you to add the pick to an existing pick.

- **Load from header:** This command can be activated from the **Tools** menu only. It allows loading the pick previously saved in the headers of the active dataset on the screen. When selecting this command, the window where you select the required header from the list, will appear.

- **Save to header:** This command can be activated from the **Tools** menu only. It allows saving the active pick into the headers of a dataset. When selecting this command the following window appears:

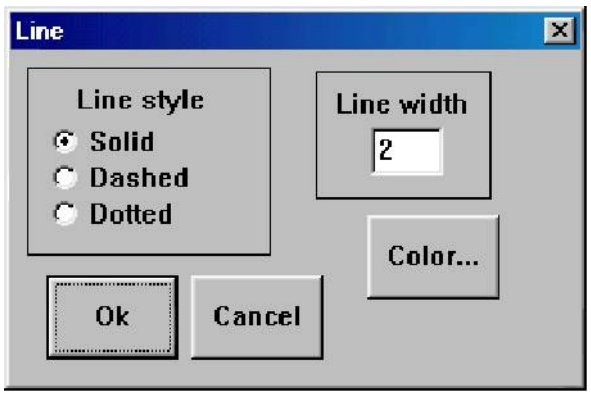
In the **Header field** field, select the header, where you desire to save the values of the active pick to. By clicking the **Reflect changes in...** you should select a dataset into which headers the pick will be saved. As a rule, you should select the same dataset where the picking has been carried out, however it is possible to pick a horizon in one dataset and save the pick into another one. The **Pick Headers...** button allows the user to select the headers to which the values of the pick will be bound.

- **Export pick:** This command can be activated from the **Tools** menu only. It allows you to save the picks as text files outside the database. When selecting this command a standard window for file saving will appear. In this window you should select the path and specify the file name.

- **Import pick:** This command can be activated from the **Tools** menu only. It allows you to load the pick from a text file saved outside the database.

- **Pick headers:** This command can be activated from the **Tools** menu. It allows viewing and changing the headers to which the pick is bound at any moment.

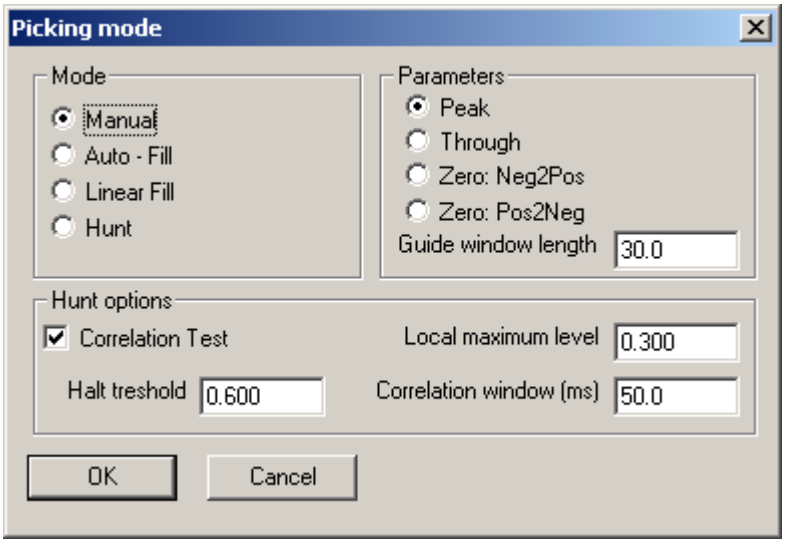
- **Line style:** This command can be activated from the **Tools** menu or by pressing the S key on the keyboard. This command makes it possible to change the style, width, and color of the active pick line. When this command is activated the following dialog box appears:



In this dialog you can select the **Line Style** (that can be either **Solid**, or **Dashed** or

Dotted) and specify the **Line Width**. The **Color** button, when clicked, opens a standard dialog window for color selection.

•**Picking mode** This command can be activated either from the **Tools** menu or by pressing the A key on the keyboard. It allows you to select the desired mode of picking and to set the appropriate parameters. . When this command is activated the following window appears:



The picking **modes** are:

•**Manual** - manual picking without any interpolation,

•**Auto - Fill** - picking with interpolation between the digitized points according to the parameter selected in the Parameters field,

•**Linear Fill** - picking with linear interpolation between the digitized points.

•**Hunt** - let the software pick automatically in both directions from a single click. Point position at the current trace will be calculated as a shift from the previous pick point determined by the maximum of normalized cross-correlation between the current trace and the previous trace within the correlation window. Picking will stop at the end of the event when the automated process fails to trace it further, i.e. when the cross-correlation within the correlation window does not have a defined maximum, or if the peak cross-correlation is found outside of the guide window, or if the quality test of the correlation fails.

The **Parameters** on the right side of the dialog are used in Auto-fill and Hunt modes only. You can specify what exactly the automated process will try to pick:

•**Peak** - picking will be accomplished according to local peak values of the signal

•**Through** - picking will be accomplished according to local minimum values of the signal

•**Zero: Neg2Pos** - picking will be accomplished according to the zero crossing from the negative values of the signal to the positive ones

•**Zero: Pos2Neg** - picking will be accomplished according to the zero crossing from the positive values of the signal to the negative ones

The **Guide window length** parameter defines the window length (in ms) where selected picking criteria (i.e. local maximum, minimum, etc.) will be searched. In fact, this value defines a maximum possible shift of the current pick point from the previous one.

At the bottom of the dialog there are additional **Hunt options** (these affect picking only in the Hunt mode):

•**Correlation Test:** this option allows testing of the quality of the correlation between the selected trace and the previous picked trace. If the maximum of the normalized cross correlation falls below the **Halt threshold**, automatic picking will be stopped. The halt threshold value can be set between 0 (no correlation) and 1 (ideal correlation).

•**Correlation window:** is the window length (in ms) where the cross-correlation function will be

calculated.

•**Local maximum level:** in the current implementation this parameter is ignored.

Pick mouse mode: This command can be activated either from the **Tools** menu or by pressing the M key on the keyboard, or, if you are in picking mode, by clicking the rectangular picking mode and horizon color indicator on the status bar. This command allows you to enter and exit the picking mode.

Marks only: This command can be activated from the **Tools** menu only. It allows switching from picking mode to trace marking mode. The traces are marked by vertical lines. All operations within marking mode are similar to those of the picking mode.

Next pick: This command can be activated either from the **Tools** menu or by pressing the TAB key on the keyboard. It switches to the next pick.

Previous pick: This command can be activated either from the **Tools** menu or by pressing the Shift-TAB keys combination on the keyboard. It switches to the previous pick.

Static Correction This menu contains commands which allow conducting static correction interactive analysis and carry out various operations with static corrections (loading, correction, saving into database, headers, as text files, importing of correction files, etc.)Static shifts can be accomplished both for one trace and for a group of traces, both simultaneously for all datasets opened in the window and for every set individually.

To select one trace or a group of traces into a block, place the cursor above the upper horizontal axis at the position corresponding to one of the edges of the selected data area and then click the left mouse button (**MB1**). Holding the button down, relocate the cursor to another edge. When the mouse button is released the selected area will be marked by color. To undo the selection, click the left mouse button (**MB1**) above the upper horizontal axis.

The selected block can be moved up/down via the keyboard arrows or the left mouse button

(**MB1**). To do this, place the cursor on the selected fragment, click the left mouse button (**MB1**) and, holding the mouse button down, move the mouse upward or downward.

Note: When using the keyboard, the selected fragment will be relocated to a value that equals the sampling step. Using the mouse makes it possible to apply static corrections less than the discrete value.

- **Simultaneous shift in panels:** If this option is activated the static shifts are accomplished simultaneously for all datasets opened in different panels. Otherwise, the static shifts are applied only to the active dataset.

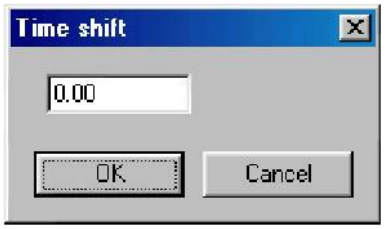
- **Clear current editing:** This command mutes either all active static shifts or a certain part. To mute only a part of the static corrections, select that part and activate the command.

- **Invert current statics:** This command inverts either all values of the specified static shifts or only a certain part. To apply the command to the part of data, select that part.

- **Load...:** This command applies static corrections previously saved in the database. When this command is activated a standard dialog box will appear. In the dialog box, select the required corrections. When doing this, the information about active corrections is displayed in the **Matching** field.

- **Save as...:** This command saves the static corrections into the database. When this command is activated, the **Save statics** diaog box, similar to the **Save pick** dialog, appears. All actions are the same as actions conducted while saving the pick. Click the **Matching field...** button of the dialog to chose a matching header field corresponding to the particular type of correction. For example, the RECNO header is chosen if the corrections are the receiver statics.

- **Export...** This command allows saving the statics as a text file outside the database. When this command is activated, first, the **Reflect matching fields** dialog appears. In this dialog, choose the required header and click OKIn the following standard file saving dialog, select the the path and the file name. **Convert to pick...** This command converts active static shifts into picks. When this command is activated the following window appears:



Here, specify the time value in ms which will be added to the statics values to obtain the pick time.

- **Save to header...:** This command saves the defined statics into the headers. When this command is activated the following window appears:

In the **Header field** field, select the header field into which the shifts values are to be saved. By clicking the **Reflect changes in...** button, select a dataset into which headers the statics will be saved. As a rule, select the dataset in which manipulations with the statics have been carried out; however, it is possible to select statics in one dataset and save them into another one. The **Reflect matching fields...** button allows the user to select the headers to which the shifts values will be bound. If any data fragment has been selected then the **Save all, Save selection** will be activated enabling to save either all statics (**Save all)** or selected fragment statics **(Save selection)** into headers. The **Save selection** is chosen by default.

- **Load from header...**This command applies statics from the headers. When this command is activated the following window appears:

In the **Header field** field from the drop-down list, choose the header for which values will be used as static shifts. This command has a number of modes:

•**Replace current** - replaces the current statics values by the statics values from the header.

•**Replace current when overlap** - instead of current statics values applies all statics values from the header that are not equal to zero. In those places where statics values from the header equal zero all current statics values do not change.

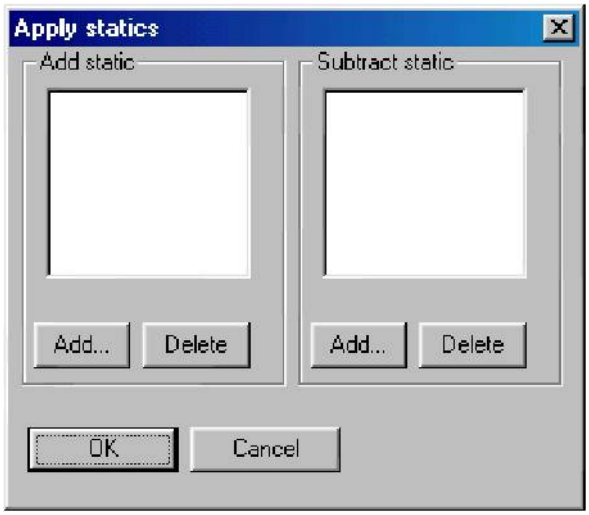
•**Add to current** - adds the statics values from the header to the current statics values.

•**Keep current when overlap** - applies static shifts values from the header only in those places where the current statics equal zero. In those places where current statics are not zero they do not change.

Convert from pick... This converts active pick into statics. When this command is activated the **Time shift** (see **Convert to pick...** chapter) window appears. Here, specify the time value in ms. The static shift value at a point of the cross-section on the screen will equal the difference between the value of the horizon (pick) time at this point and the numeral specified in the **Time shift** window.

Difference between picks: This command enables calculation of statics as the difference of values between two picks. This command is available if there are only two different picks in the window.

Apply statics...: This command applies the statics from headers that will not be considered current statics for the data. When this command is activated the following window appears:



In the **Add statics field** from the list launched by clicking the **Add...** button, select the required header. The values it contains will be applied as statics to the data. The **Subtract statics** field is used when you have to subtract some other statics values saved in headers from values of statics selected in the **Add statics** window. The **Delete** option is used to delete the selected header from the **Add statics** field or from the **Subtract statics** field.

Apply procedure: When this menu item is selected the sub-menu containing the list of available processing routings which can be applied to the data directly in the *Screen Display* module will open. When using the processing routines inside the *Screen Display* module, all changes will be saved into the data passing through the flow. I.e. further, after the module window has been closed, the processed data will pass through the flow. To obtain reference information about specific routines in this menu, see the description of similar processing modules.

Amplitude editing: This menu item is used for various manipulations with amplitudes, such as gain/attenuation, and polarity inversion.

In this module it is possible to carry out data gain control manually. To do this, the traces should be combined in a block. When the "+" or "-" keys on the keyboard are pressed, the amplitudes of the selected traces will be multiplied or divided by 1.4, respectively. This gain/attenuation affects the data display on the screen only.

Manual gain coefficients will be saved together with the flow and will be recovered when starting the flow every time.

Cancel amplitude editing: This command allows the user to cancel all manually conducted operations with amplitudes.

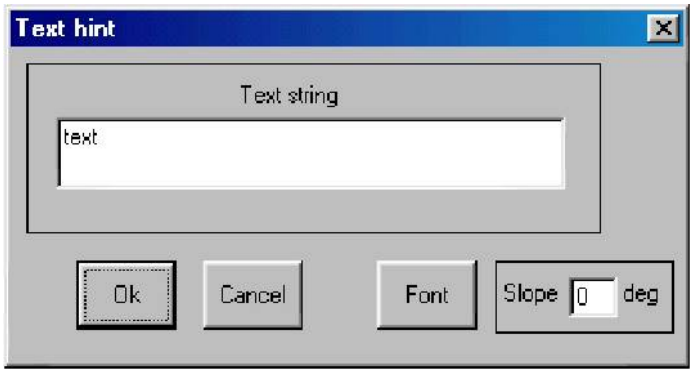
Invert traces: This command allows the user to accomplish polarity inversion for a selected group of traces.

Load from header: This command allows the user to apply multiplication of coefficients saved in the headers to current amplitudes. The header should be selected in a standard window that appears when this command is activated.

Save to header: This command allows the user to save current, manually specified, coefficients on which the amplitudes have been multiplied, into headers. The header should be selected in a standard window that appears when this command is activated.

Trace header math...: This command accomplishes various mathematical operations with headers. See the description of *Trace Header Math* module in the **Geometry/Headers** chapter.

Text hint...: This command allows creating a text string label placed in the working area. When this command is activated the following window appears:



In the **Text string** field, enter the label text. The **Font** button selects the font, its style and size. In the **Slope** field, specify the text string slope expressed in degrees (counter-clockwise).

To relocate the created label on the screen, capture it with the right mouse button (**MB2**) and relocate it to a new place.

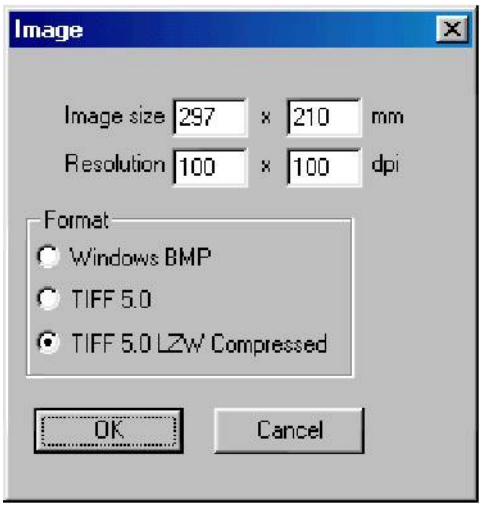
To delete the label, double-click the right mouse button (**MB2**) on the lable.

To change the label parameters (content, font, slope), click the left mouse button (**MB1**) on the label.

Print...: This command prints out the data.

Print options...: This command prints parameters setting.

Save image...: This command saves the running window content as a bitmap image on disk. When this command is activated the window for image parameters settings will open:



In the **Image size** fields the image size should be specified in mm, in the **Resolution** field the image resolution should be specified in dots per inch. In the **Format** field, define the format of a file in which the image will be saved.

NOTE: in the *Windows BMP* format, the information about resolution will not be saved. When correct image size is important it is recommended to use the TIFF format. Besides, TIFF files with LZW compression are usually much smaller than files in BMP format.

- **Exit:** Exits the ***Screen Display*** module.

- **Help:** This command launches the help for the program.

Plotting*

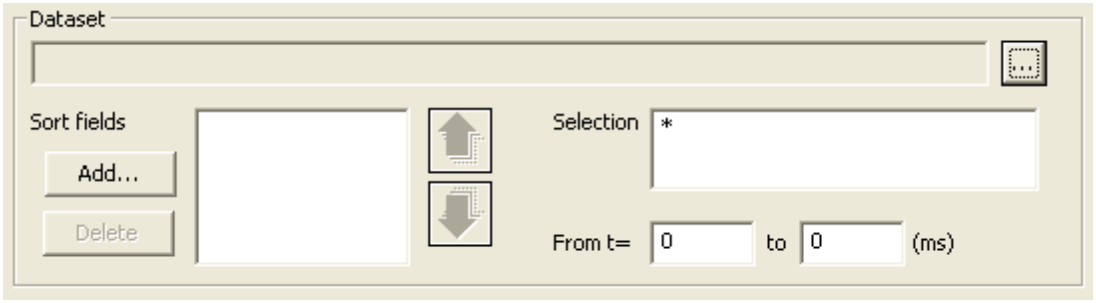
This module outputs processing results to any printing device compatible with Windows operating system. The module allows changing data visualization parameters (sorting, display method,

scaling, amplification, pick and header plot visualization, line width, font size etc.), adding labels and logos to the image, and working with all standard print setup functions (including image preview before printing).

Plotting is a so-called standalone module, i.e. a module that does not require other modules in the flow.

Plotting parameters

The **Dataset** field allows the user to select a dataset that shall serve as a source for generation of the image to be printed. When the button is clicked, a dialog box will appear prompting the user to select a dataset from the project database.



The **Add…** button in the **Sort Fields** field opens a standard dialog box containing a header field selection list. The user should select appropriate input data sorting keys from this list. The selected keys will appear in the **Sort Fields** list. Keys can be moved up and down in the list using the arrow buttons to the right of the list. Keys can also be deleted from the list by pressing the **Delete** button.

The **Selection** field allows specifying the input range for each key. Different key ranges are separated with a colon.

Example:

Let us assume that two sorting keys are selected in the **Sort Fields** field. In this case, the range string in the **Selection** field may look like the following:

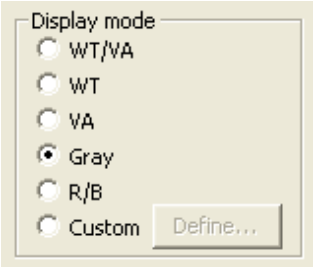
: – all data sorted in accordance with the two selected keys will be input.

*:1000-2000(5) – data for the second key will be input in the 1000 to 2000 range at increments of 5.

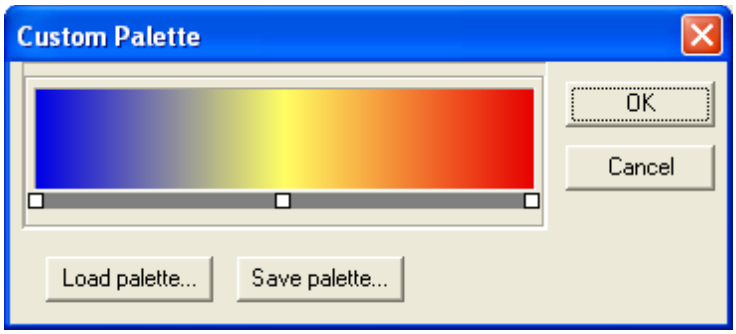
From t= – start time in ms. Time sections will be displayed on the preview screen and printed starting from this time.

to – end time in ms. Time sections will be displayed on the preview screen and printed down to this time. To display all samples down to the end of the trace, enter "0" in this field.

Display mode



- **WT/VA** – displays the traces using the wiggle trace/variable area method;
- **WT** – displays the traces using the wiggle trace method;
- **VA** – displays the traces using the variable area method;
- **Gray** – displays the traces using the variable density method in the gray-scale palette;
- **R/B** – displays the traces using the variable density method in the red-white-blue palette;
- **Custom** – displays the traces using the variable density method in a custom palette,
- **Define** – this button is active if the **Custom** option is enabled. Pressing the button opens the **Custom Palette** window.



The palette is specified as a set of points with assigned colors. Colors are interpolated linearly between the specified points. The user can create, move and delete points and change their assigned colors. Points are shown as white rectangles on the gray strip under the palette image.

- To **move a point,** select and drag it with the left mouse button **(MB1)**. The palette will change its appearance as you move the point.

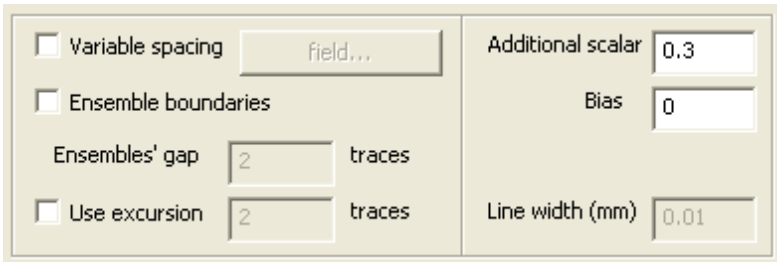
- To **change the color assigned to an existing point,** double-click it with the left mouse button **(MB1)**. A standard point color selection dialog box will appear.

- To **create a new point** with a specific color assigned to it, Shift-click the appropriate spot on the palette using the left mouse button **(Shift+MB1)**. A standard point color selection dialog box will appear. A new point with the selected color will be added in the location you clicked on.

- To remove a point from the palette, click on it using the right mouse button **(MB2)**.

- **Load palette...** Pressing this button opens a standard file opening dialog box to select an RGB ASCII text file from the disk;

- **Save palette ...** Pressing this button opens a standard file saving dialog box to save the current palette to the disk as an RGB ASCII text file.



- **Variable spacing**. Select this option to arrange the traces on the preview screen/plot in accordance with the specified header values. For example, this option allows arranging stacked section traces on the screen by their world coordinate along the profile. To select a header, press the **field** button.

- **Ensemble boundaries**. Enable this option to have the ***Plotting*** separate different trace ensembles on the preview screen/plot with gaps. The first sorting key specified in the stream data input module is the ensemble key.

- **Ensemble's gap**. Gap width (specified as the number of traces) between trace ensembles. The field is active if the **Ensemble boundaries** option is enabled.

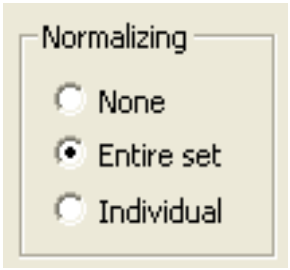
- **Use excursion ___ traces**. This option is used to limit the maximum trace display deviation when using the WT, VA or WT/VA method. If this option is enabled, amplitudes exceeding the specified maximum deviation will be clipped on the screen when the trace is displayed.

- **Additional scalar**. (Display gain). An additional factor to multiply trace samples by before being displayed on the screen/plot.

- **Bias**. Trace mean level shift relative to the zero. If traces are displayed using the variable area method, changing this parameter leads to a change in the black level. Positive values will result in a shift to the left from the trace zero line and an increase in the blackened area of the curve. Negative values will reduce the blackened area of the curve. If data is displayed using any of the variable density methods (Gray, R/B or Custom), this value will shift the zero relative to the palette center.

- **Line width (mm)** – width of lines displayed on the screen. This option can be used when traces are displayed using the WT/VA or WT method.

Normalizing (normalization of trace amplitudes displayed on the screen/plotted)

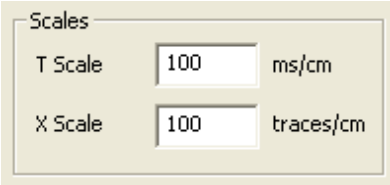


- **None** – no additional trace normalizing;

- **Entire screen** – normalizes all traces by dividing trace amplitude by the average absolute amplitude value of all traces;

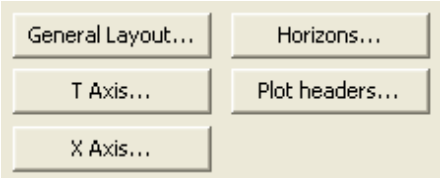
- **Individual** – normalizes each trace individually by dividing trace amplitude by the average absolute amplitude value of the trace itself.

Scales



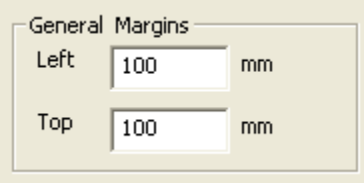
- **TScale**. Vertical scale value in ms/cm.

- **Xscale**. Horizontal scale value in traces/cm.



General Layout… (label and margin setup)

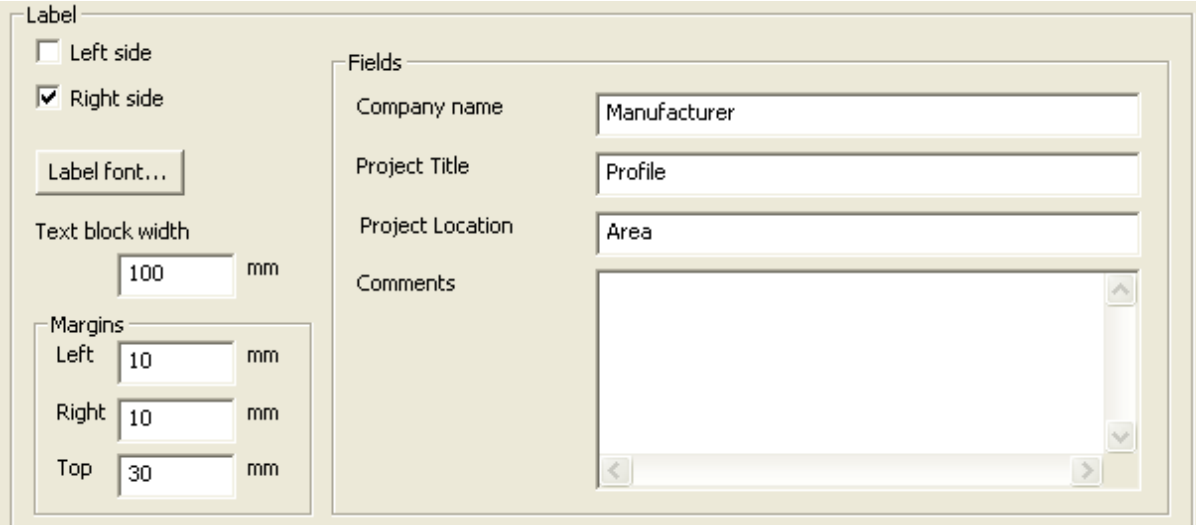
General Margins (margin width)



- **Left** – image margin from the left edge of the sheet in mm**;**

- **Top –** image margin from the top edge of the sheet in mm.

Label



- **Left side** – plot label to the left of the image;
- **Right side –** plot label to the right of the image;
- **Label font**… – Click this button to opens the label text font setup dialog box;
- **Text block width** – text block width in mm.
- **Margins**

Left – label margin from the left edge of the sheet in mm;

Right – label margin from the right edge of the sheet in mm;

Top – label margin from the top edge of the sheet in mm.

- **Fields** - label text fields are as following:

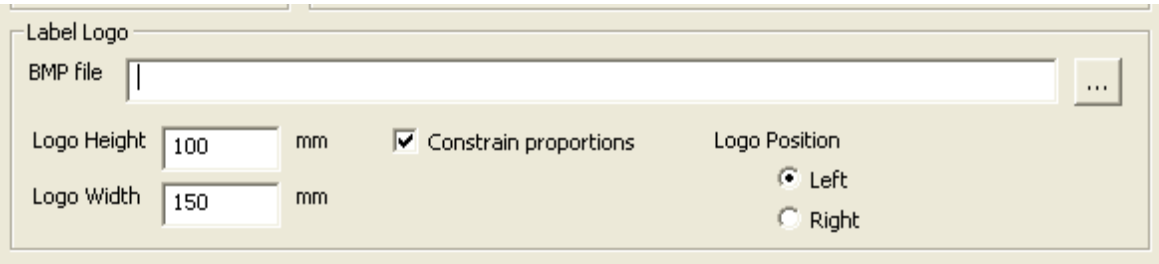
Company name;

Project Title;

Project Location;

- **Comments** – for any additional information.

Label logo

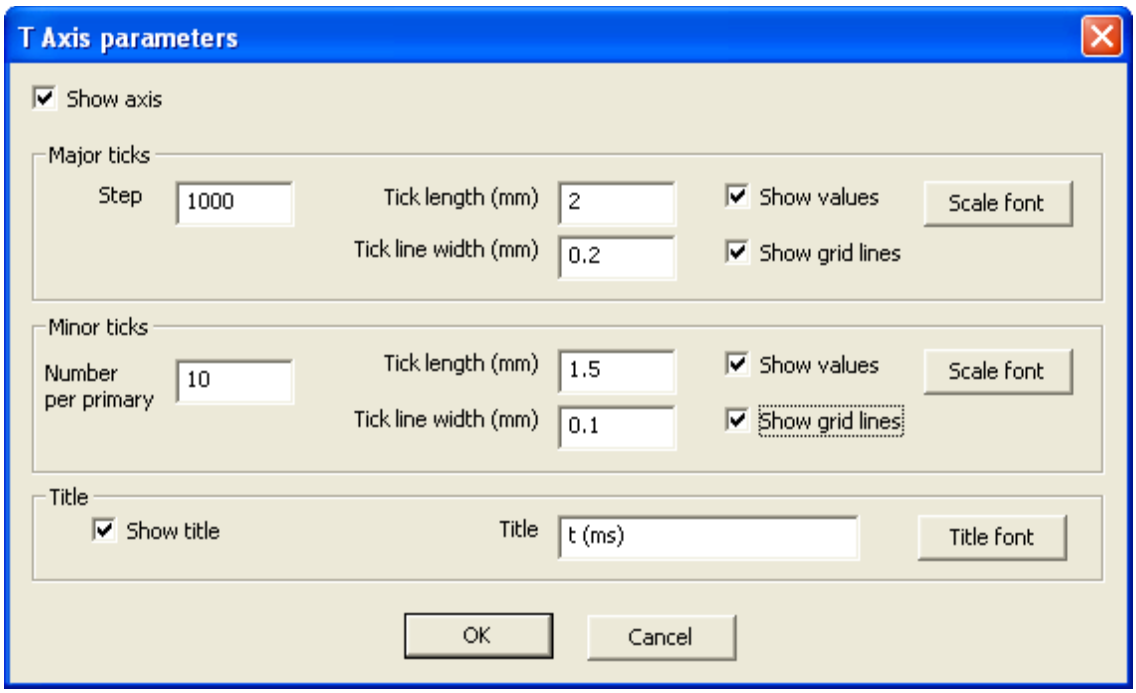


The **BMP file (image in *.bmp format)** field allows the user to choose a logo to be added to the label. Clicking the button will open a file selection dialog box.

- **Logo Height** – logo height in mm;
- **Logo Width** – logo width in mm;
- **Constrain proportions** – preserve/do not preserve initial proportions.
- Logo Position

- **Left** – in the left part of the label;
- **Right** – in the right part of the label.

T Axis…(time axis parameters)



- **Show axis** – show/do not show axis titles and scale ticks

Enabling **Show axis** allows the user to specify major and minor tick display parameters and their value marks as well as axis titles.

- **Major ticks**

Step – tick interval in ms;

Tick length (mm);

Tick line width (mm);

Show values – show/do not show scale values;

Show grid lines – show/do not show grid lines;

Scale font – clicking this button opens the major tick value mark font setup dialog box.

- Minor ticks

- **Number per primary** – number of minor ticks per one major tick;

- All other parameters are identical to **Major ticks** parameters**.**

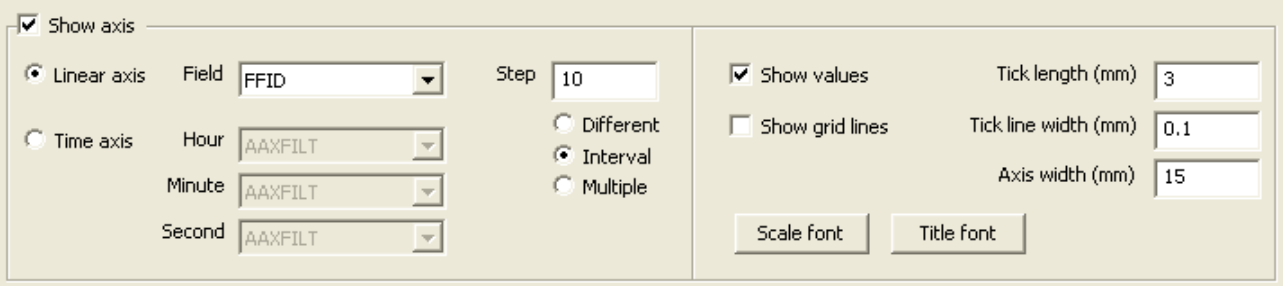
- **Title**

- **Show title** – show/do not show axis title;

- **Title** – title text;

- **Title font** – clicking this button opens the axis title font setup dialog box.

X Axis parameters (horizontal axis parameters)



This group consists of three identical fields used to specify the parameters of horizontal scale titles – trace labels. Several trace labels can be used in this case, with each label set up independently.

- **Show axis** – show/do not show axis titles and scale ticks;

- **Linear axis** – linear axes, displays the values of the specified header field.

- To select a header field whose values will be used as a trace label, press the **Field** button.

- **Time axis** – displays shooting time stamps (hour, minute and second).

- If header fields containing hours, minutes and seconds are completed in the project, they can be selected in the appropriate **Hour**, **Minute,** and **Second** fields and used as axis titles.

- It is possible to specify the trace title step, i.e. interval between trace labels appearing on the horizontal scale **(Step)**. Possible options:

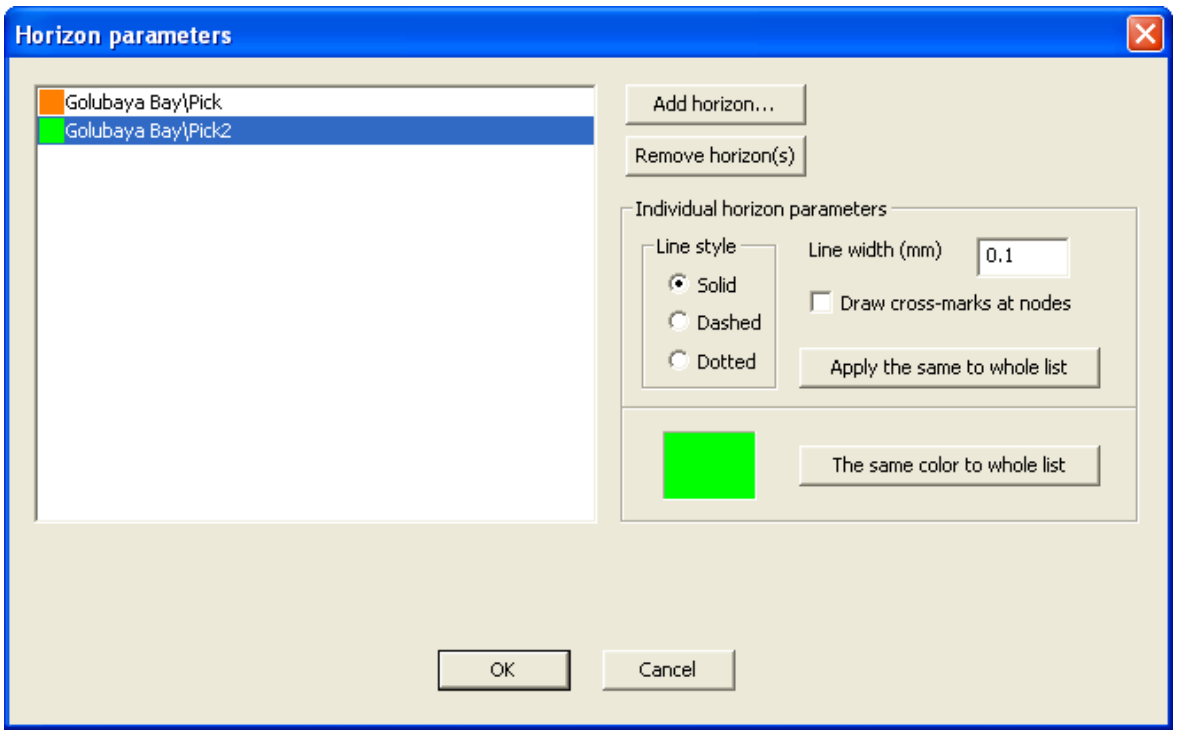
Different shows a title for the first trace and every subsequent trace with header value different from the previous trace (does not work for the **Time axis**);

Interval shows a title for the first trace and each N^{th} trace after it. Specify the desired N interval in the **Step** field;

Multiple shows a title for the trace if its header value is a multiple of the increment specified in the **Step** field.

- **Show values** – show/do not show scale values;
- **Show grid lines** – show/do not show grid lines;
- **Tick length (mm)** – tick length in mm;
- **Tick line width (mm)** – tick line width in mm;
- **Axis width (mm)** – axis width in mm;
- **Scale font** – click this button to open the scale tick value mark font setup dialog box;
- **Title font** – click this button to open the axis title font dialog box.

Horizons… (display of horizon pick)



This option allows setting up the pick display parameters. A list of displayed picks is shown on the left side of the setup dialog box. Picks may be added to the list by the **Add horizon…**button or deleted from the list by the **Remove horizon(s)** button. To select multiple picks to be removed, Shift-click on them using the left mouse button (MB1).

- Individual horizon parameters

- **Line style** can be the following:

Solid,

Dashed,

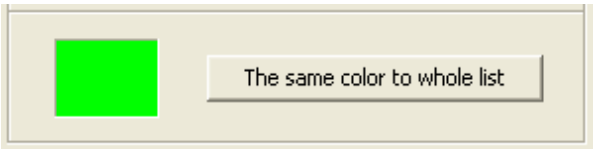
Dotted.

- **Line width (mm)** – line width in mm;

- **Draw cross-marks at nodes** – draw pick nodes;

- **Apply the same to whole list** – apply the current pick visualization settings to the entire list of

picks.

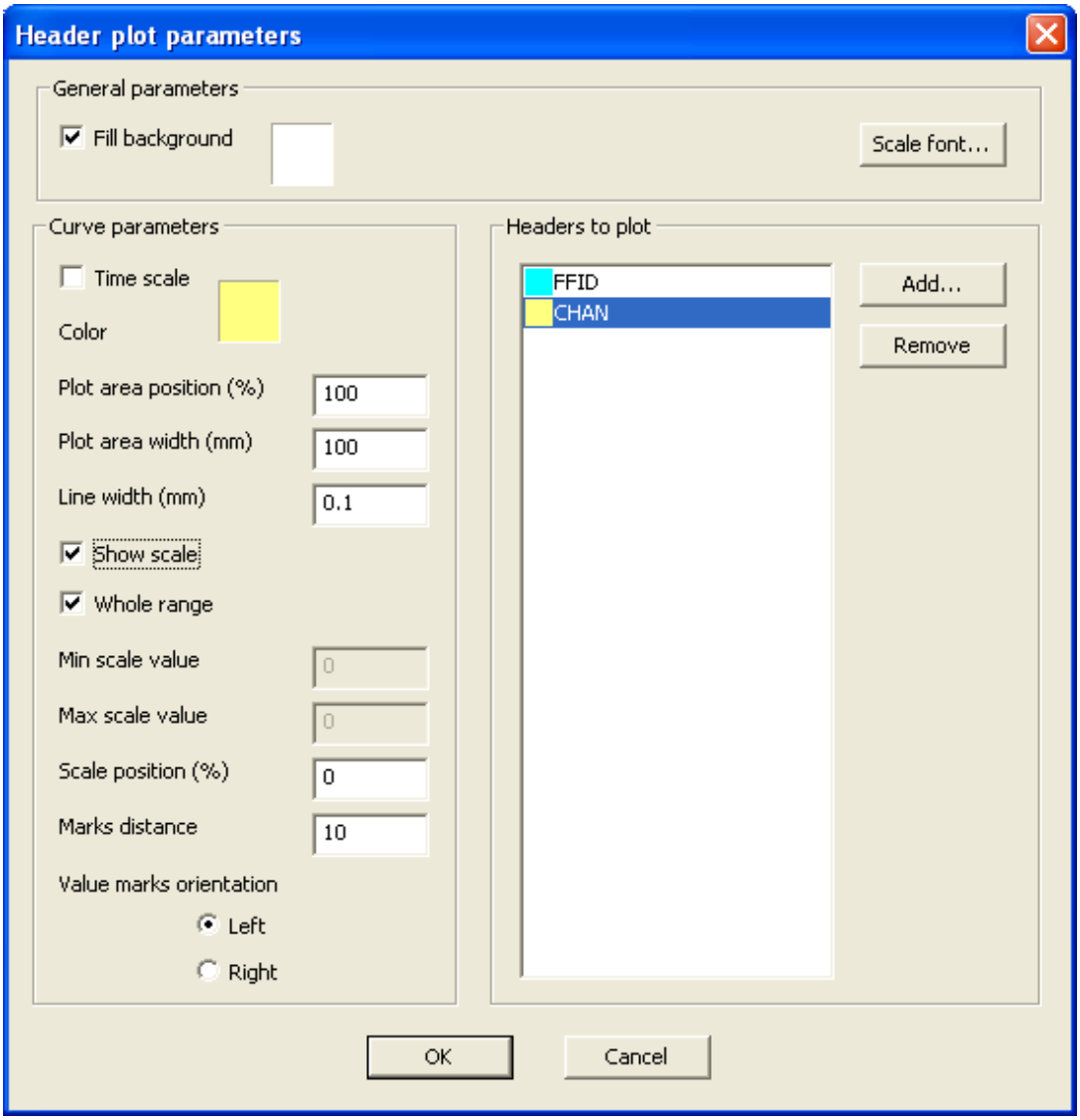


- This field allows selecting the pick color. To open the color list and change the pick color, click the color square with the left mouse button and select a new pick color from the list.

- **The same color to whole list –** apply the current color to all picks in the list

Plot headers…

Pressing this button opens the **Header plot parameters** dialog box that allows specifying the preview screen/plot display parameters for selected header field value plots:



- General parameters. This is a group of general parameters common for all curves.

Fill background – this option is used to display plots against a solid background. Click on the colored square to the right to select the background color.

Scale font… – clicking this button opens the plot tick font setup dialog box.

- **Headers to plot** – this option is used to select the headers to be displayed as plots.

Add. Use this button to select the header fields to be displayed. A list of headers available in the current project will open. You can select several headers from the list by left-clicking while pressing the Shift and Ctrl keys. After selecting the headers, press the **OK** button. The selected headers will appear in the list.

Remove. This button removes the selected header fields from the displayed plot list. You can select several headers in the list by left-clicking while holding down the Shift and Ctrl keys.

- **Curve parameters.** This section is used to set up individual display parameters for each header field in the list. Left-click on one of the selected header fields in the list to view and change its display parameters.

Time scale. Enable this option to have header fields interpreted as time values and displayed in accordance with the existing time scale. This option, for example, can be used to print static shift values in the real scale.

Color. Specify the display color for the selected header field here. Click on the colored square to select a new color.

The fields below are used to set up the plot display scale. They are active only if the **Time scale** option is enabled.

Plot area position (%). Specify header plot downward shift as percentage of the image size. If the value of this field is set to “0”, the plot will be displayed immediately above the time section, if to “100” - at the bottom of the section.

Plot area width (mm). Height of the area (in mm) in which the selected header field value plot will be displayed.

Line width (mm) – line width in mm.

Whole range. When this option is enabled, the entire range of header values is scaled to fit it completely into the specified display area. Otherwise the displayed value range should be set manually by specifying its minimum (**Min scale value**) and maximum (**Max scale value**) values.

Show scale. This option allows displaying an axis with a step scale for the selected plot. If it is enabled, the axis position can be specified in the **Scale Position** field as percentage of the screen width (0 – left edge, 100 – right edge).

Value marks orientation allows specifying the value mark position relative to the scale axis – to the left (**Left**) or to the right (**Right**).

After setting up plot display parameters for one of the header fields in the list, left-click another one. The parameters specified for the previous field will be saved. After setting up all necessary parameters, press the **OK** button. Selected header field values will be displayed as plots in accordance with the specified parameters.

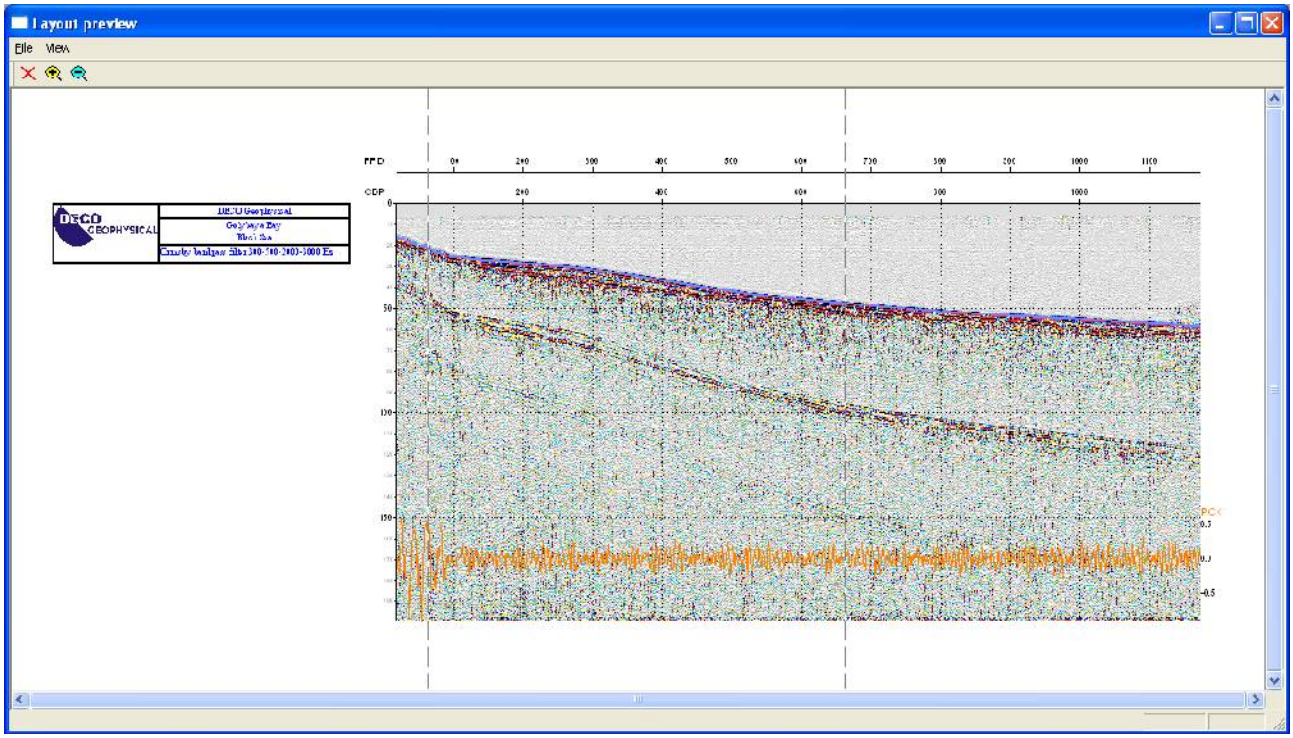
Image preview and print setup

The **Print setup…** option allows selecting a printer and adjusting its parameters in a standard Windows print setup dialog box.

The **Display traces in Layout Preview** option allows selecting whether to display seismic traces in the preview window (it is recommended to switch it off for big lines, otherwise the preview will be very slow).

The **Layout Preview**… option allows previewing the image before printing. Clicking the

button opens a print preview window (**Layout preview**).



This window allows the user to preview the image in general, check how many sheet the image will be printed on, and zoom in or out by pressing the buttons or selecting the **View/Zoom In, View/Zoom Out** from the menu. To close the window, press the button or select **File/Exit** from the menu**.**

To print the image, set all necessary parameters in the **Plotting parameters** dialog box and click **OK**. Then use the **Run** command of the flow editor to execute the flow with the module.

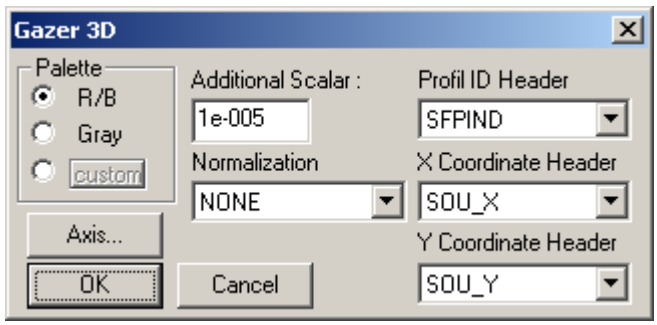
3D Gazer (Display of seismic crossections in real 3D coordinates)

The module is dedicated to 3D display of the seismic srossections along the lines. The crossections are displayed in their real 3D coordinates.

Each line at the module input must have a unique line ID value stored in one of the trace header field (by default, SFPIND). This field must be indicated as the first sorting key in the **Trace Input** module when entering the traces from several datasets into the flow using one single instance of the **Trace Input** (in order to ensure that the traces from one line goes together).

Module parameters

The parameter dialogue looks as following:



- You can select colour **Palette** that will be used for data display. The following options are available:

- **R/B** – red-white-blue palette;

- **Gray** – greyscale palette.

- **Custom** – when this option is on, the corresponding button calls palette secection dialogue similar to that used in the **Screen display** routine.

- **Additional Scalar:** - screen gain.

- **Normalization** – allows selecting amplitude normalization method. Available options include: NONE – no normalization applied, ENTIRE – all traces are normalized altogether, INDIVIDUAL – each trace is normalized separately.

- **Axis** – this button calles the dialogue with axes parameters. You can indicate intervals for primary and secondary ticks separately for the time and coordinates. It is also possible to adjust the font.

- **Profile ID header** – trace header field with the unique line ID value.

- **X Coordinate Header** - trace header field with X-coordinate.

- **Y Coordinate Header** - trace header field with Y-coordinate.

Working with the module

After the module is launched you will see its main working window (Fig.1) containing 3D image of the loaded data and coordinate axes. Horizontal exes display coordinates in meters, vertical axis – two-way time in nanoseconds.

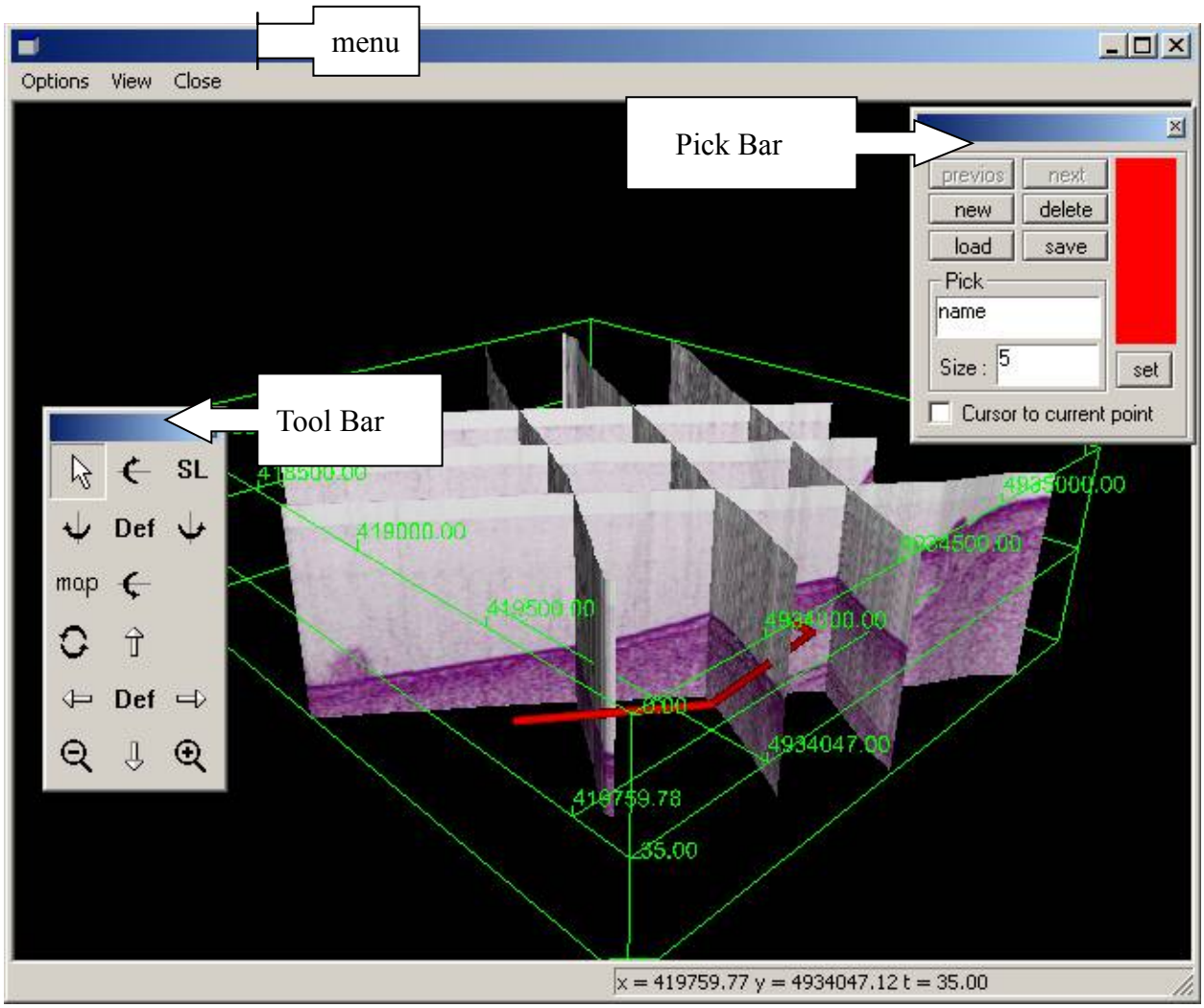


Fig. 1. Main working window with available bars.

Besides, 2 special bars are available: the **Tool Bar** (by default is visible) and the **Pick Bar** (by default is invisible). You can switch the bars on and off through the **View** menu.

Controlling 3D scene

Tool Bar

The **Tool Bar** (Fig.2.) is designed to control the position of the 3D scene relative to the user, as well as to switch on and off some additional working modes. The purpose of the command buttons is described below:

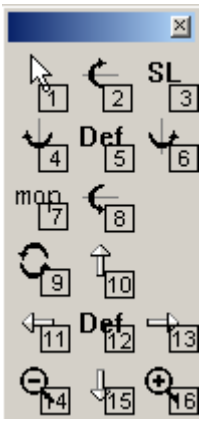


Fig.2

- Buttons (2),(4),(6),(8) allow rotating the scene relative to two mutually-perpendicular axes, button (5) brings the scene back into default position.

- Buttons (10),(11),(13),(15) allow moving the scene left, right, up and downd, button (12) brings the scene back into default position.

- Buttons (14) and (16) allows zooming and unzooming the scene.

- Button (1) switches on/off the Cursor-mode (see details about this mode in *"Working in Cursor-mode"* section).

- Button (3) switches on/off Slice-mode. In this mode, the amplitude slices of all the profiles are displayed (Fig. 3). The width of slice display can be adjusted through the menu (**Options/Slice Properties…)**. Vertical position of the slices can be controlled by "Up" and "Down" arrow-keys, or (when Cursor-mode is on) by mouse.

- Button (7) switches on/off displaying of survey map above the profiles. (A map must be previously loaded by **Options/Map…**, see corresponding section n of this manual).

- Button (9) switches on/off continuous rotation mode. In this mode, rotating the csene by mose would cause ints continuous rotation in the specified direction.

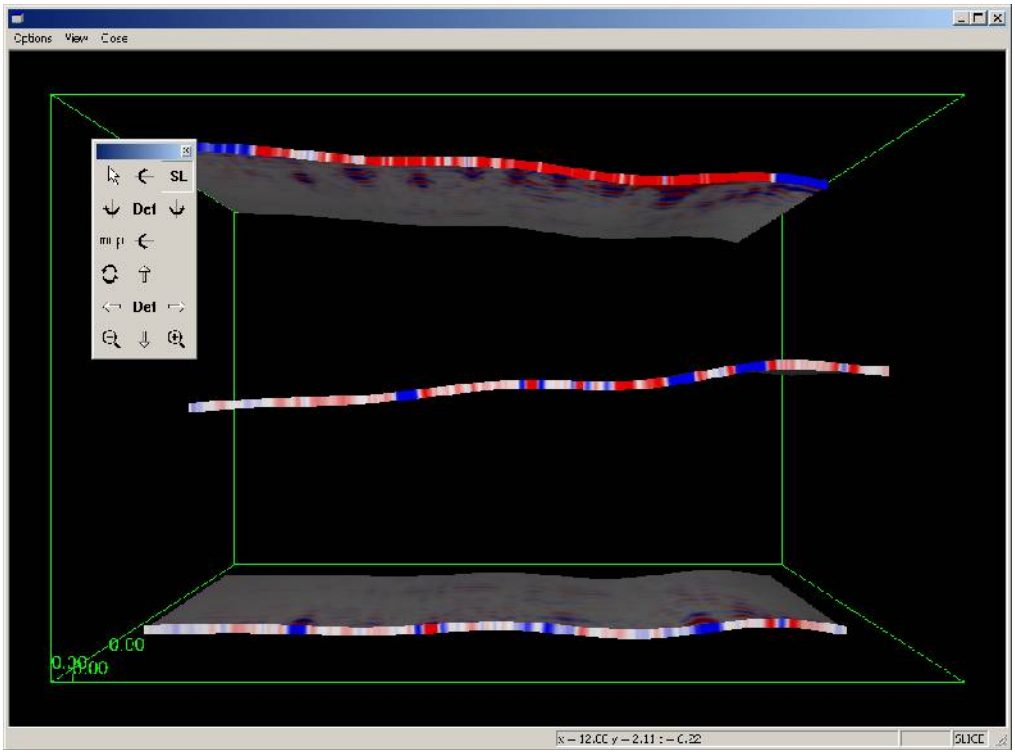


Fig. 3. Slice-mode.

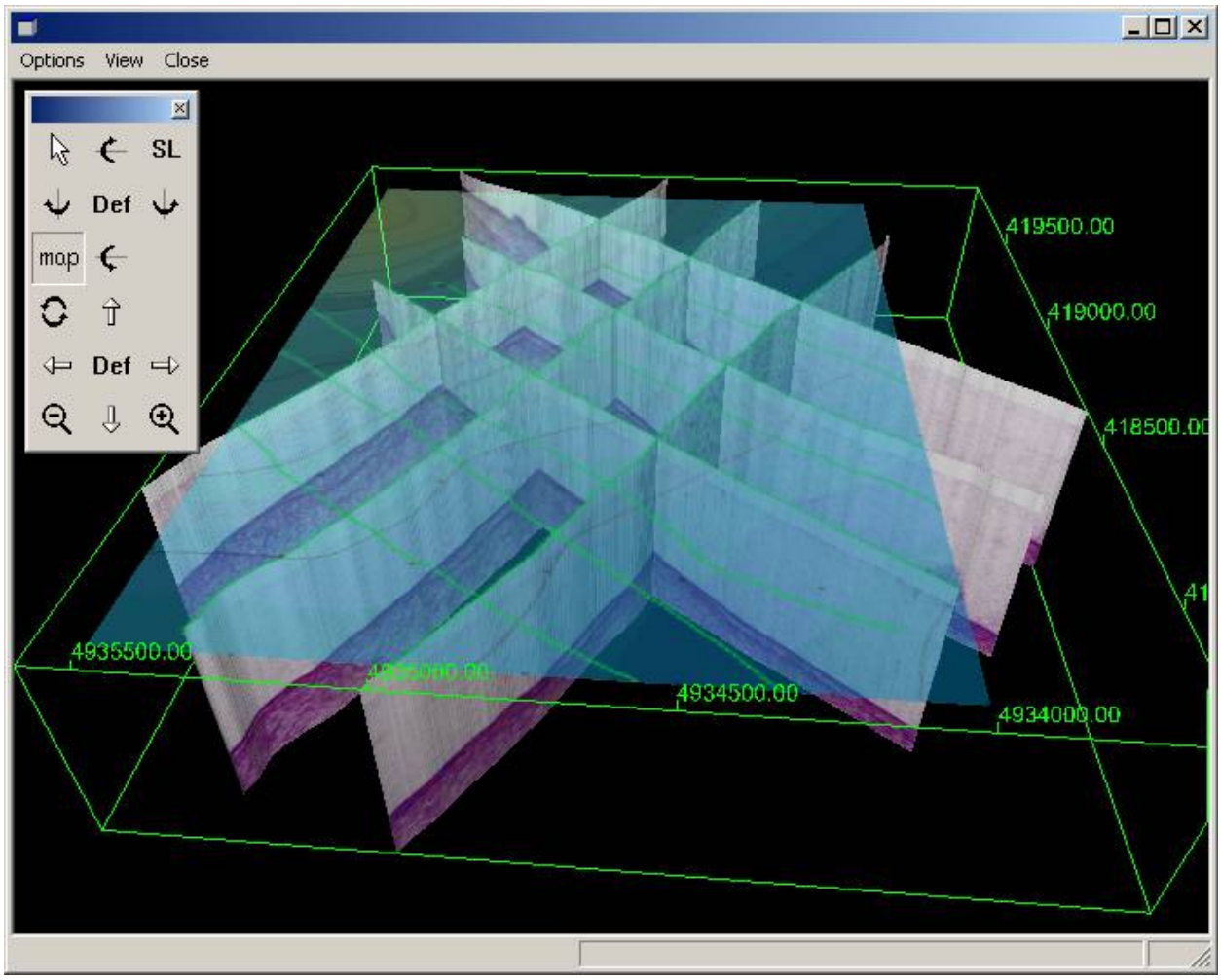


Fig.4. Viewing map above the profiles

Controlling the scene with the mouse and keyboard in the normal mode

In 3D Gazer you can work with the mouse in two modes: normal-mode and cursor-mode.

In normal mode you can use the mouse to rotate the scene, move it, zoom up and zoom out.

For **rotating** the scene, click left mouse button at any point of the screen and while holding the button move the mouse cursor. The scene will be rotated in the specified direction.

For **moving** the scene relative to the observer move the mouse cursor holding its right buton pressed.

For **zooming/unzooming** the scene hold left mouse button and move the mouse cursor up/down keeping Ctrl key pressed.

The keyboard can be used to **change the vertical scale** of the scene – use Ctrl+"Up" and Ctrl+"Down" key combinations

Working in Cursor mode

In the Curor mode a 3D cursor is displayed on the screen. It can be used to determine coordinates of any point on the screen (Fig. 5). X and Y coordinates as well as two-way time (t) in nanoseconds are displayed along the coordinate axes and also at the status bar.

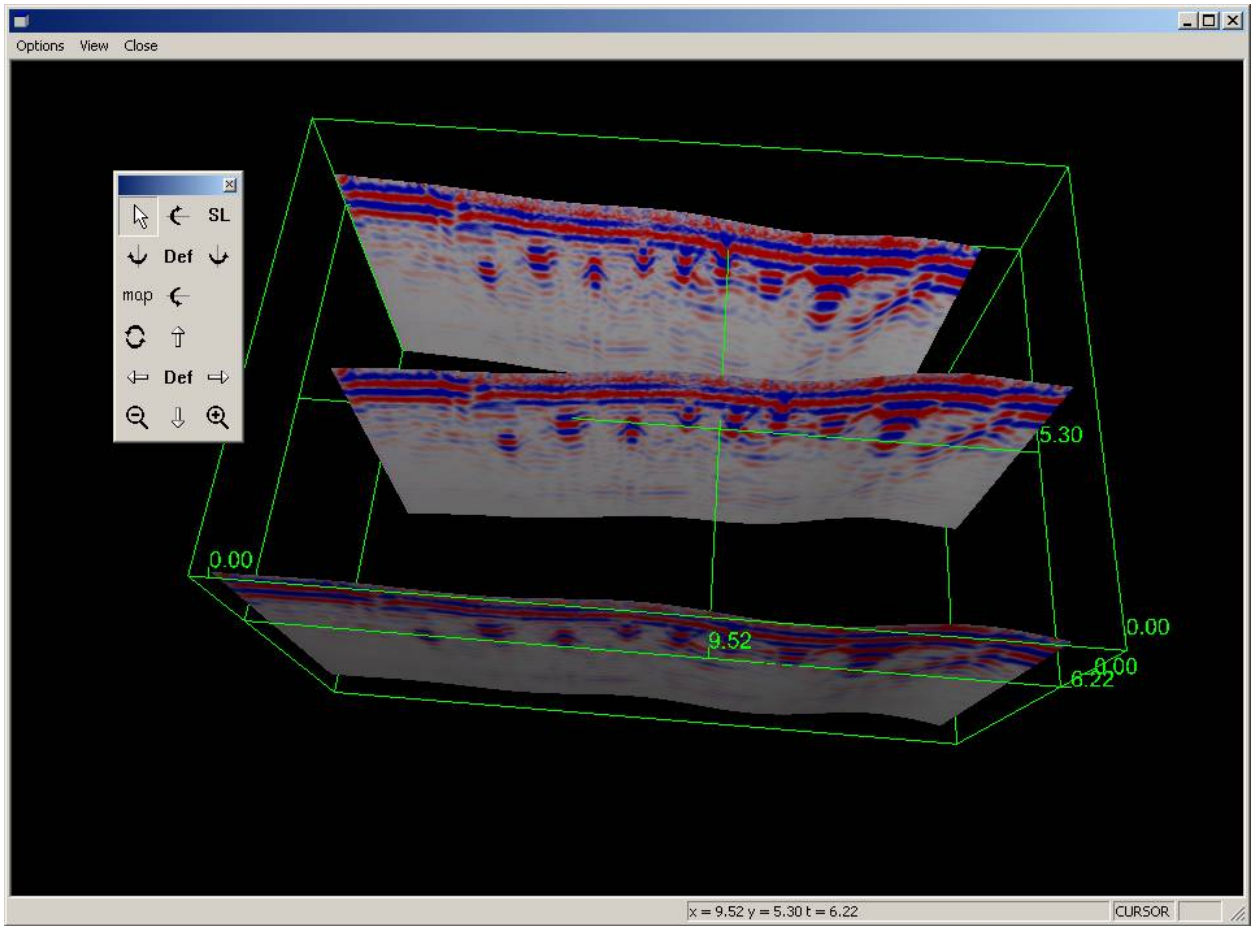


Fig. 5. Cursor mode.

In Cursor-mode the mouse controls *only* the 3D cursor position. You can control the scene position through the **Tool Bar**.

When left mouse button is pressed, **3D cursor will follow the mouse cursor in XY-plane**. The vertical cursor position (along time axis) does not change at that.

When the left mouse is pressed together with the Shift key, **moving the mouse cursor will change the vertical position of the 3D cursor**. At that, the position of the 3D cursor in XY-plane will not change.

Picks

Pick Bar

You can work with picks through the **Pick Bar** (Fig.6). To display the **Pick Bar** on the screen, use **View/Pick Bar…** menu command.

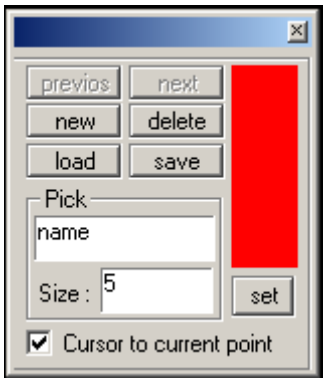


Fig. 6. Pick bar.

On the pick bar the **new** button creates a new pick, **delete** button deletes the current pick. With the corresponding buttons you can also **save** current pick into a text file and **load** some previously saved pick on the screen.

IMPORTANT: Since the 3D Gazer picks correspond to linear objects rather than to horizons, their file format is different from the **RadExPro** horizon picks. XY coordinates of pick nodes and the node time values are stored together with the line style and colour information

You can switch from one pick to another with **previos** and **next** buttons.

The colored rectangular field to the right shows the color of the current (active) pick. Click on this field to select a new coulour for the pick in a standard color-select dialog.

In the **Pick** group of fields you can specify the name of the current pick and its **Size**.

Set button updates parameters of the active pick in accordance with the current pick bar settings.

Option of automatic cursor positioning to current pick points (**Cursor to current point)**, allows in the Cursor-mode to quickly move 3D cursor from one pick point to another. For that, switch the option on, use the tool bar to switch on the Cursor-mode, and then use Ctrl+"Right" and Ctrl+"Left" key combinations. The 3D cursor will jump sequentially from one current pick point to another.

Editing picks on the screen

For editing picks on the screen you need to switch to the Cursor-mode.

To **add a point** to the current pick, position the 3D cursor to the desired location of the 3D scene, place the mouse cursor to the same location and click the left mouse button holding Ctrl key pressed. A point can be added either to the end or before the beginning of the pick. To make this, before adding a point, “illuminate” either the first or the last point in the pick (a pick point is “illuminated” when a 3D cursor is placed on the corresponding position in the 3D scene, see fig. 7). A point can also be added between two existing pick points. To make this, position the 3D cursor to the connecting line, add a point on the line and then move it to the desired location.

To **move a point** of the current pick, position the 3D cursor at the desired point (when the cursor is properly positioned at a pick point, the point becomes “illuminated”, see fig.7), place the mouse cursor to the same location of the screen, then catch the point with the right mouse button and drag the point to the new position (to move the point along the time-axis, keep Shif key pressed).

To **delete a point** of the active pick, position the 3D cursor at the desired point (when the cursor is properly positioned at a pick point, the point becomes “illuminated”, see fig.7), place the mouse cursor to the same location of the screen, then double-click the point with the right-mouse button.

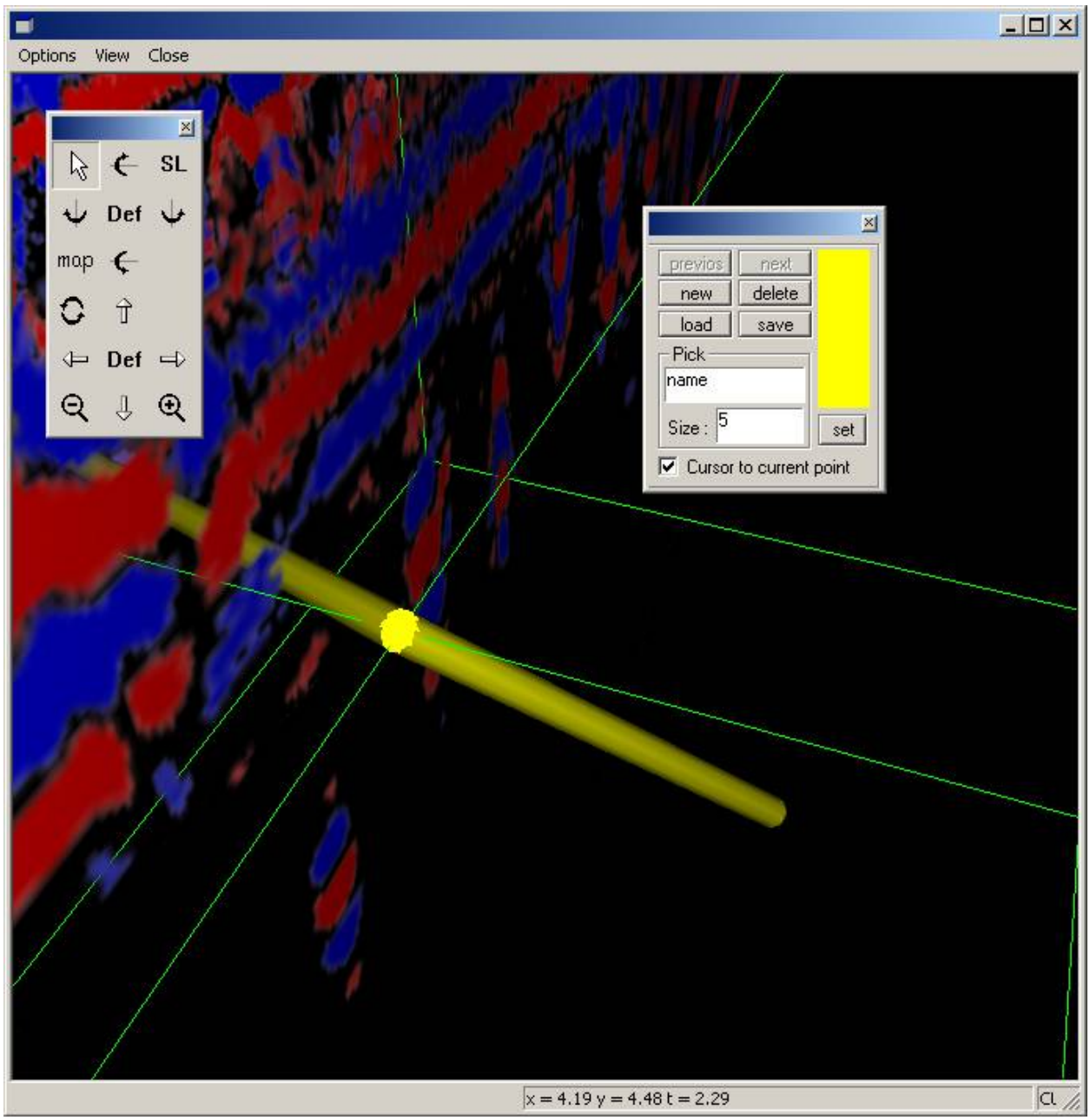


Fig. 7. Editing pick on the screen: when 3D cursor is positioned at a pick point, the point is "illuminated".

TIP: When moving/deleting an existing point of the active pick it is convenient to use the option of automatic cursor positioning to current pick points (**Cursor to current point)** of the Pick Bar.

TIP: When adding/editing pick points it is often convenient to switch on slice-mode, as in this mode positioning of 3D cursor appears to be more simple and obvious.

Adjusting data display parameters

Adjustment of data display parameters can be made through the Display Properties dialog window that is called by **Options/Display Properties** menu command (Fig. 8).

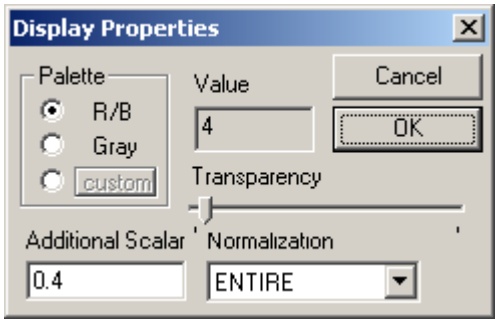


Fig. 8. Data display parameter dialog

You can select color **Palette** that will be used for data display. Available options are as following:

R/B – red-white-blue

Grey – grey-scale

Custom – user-defined color palette.

When user-defined palette is selected, the **Custom** becomes enabled. The button is used for selecting/editing custom color palettes. When the button is pressed, the palette dialog will appear, similar to that used in the *Screen Display* module. The dialog allows editing the palette, saving it to hard disk (**Save** button) and loading previously saved palettes from disk (**Load** button).

Additional Scalar is a screen gain coefficient value. **Normalization** drop-down list allows setting a mode of amplitude normalization prior to data display (ENTIRE – one and the same normalizing factor is calculated for all traces, INDIVIDUAL – separate normalizing factor is calculated for every individual trace, NONE – no additional normalization is performed).

Transparency slider allows making some portion of low-amplitudes transparent, i.e. to display only the strong amplitude parts of the sections. (Fig. 9). The current transparency value (the percentage

of absolute amplitude values that become transparent) is reflected above the slider in the **Value** field.

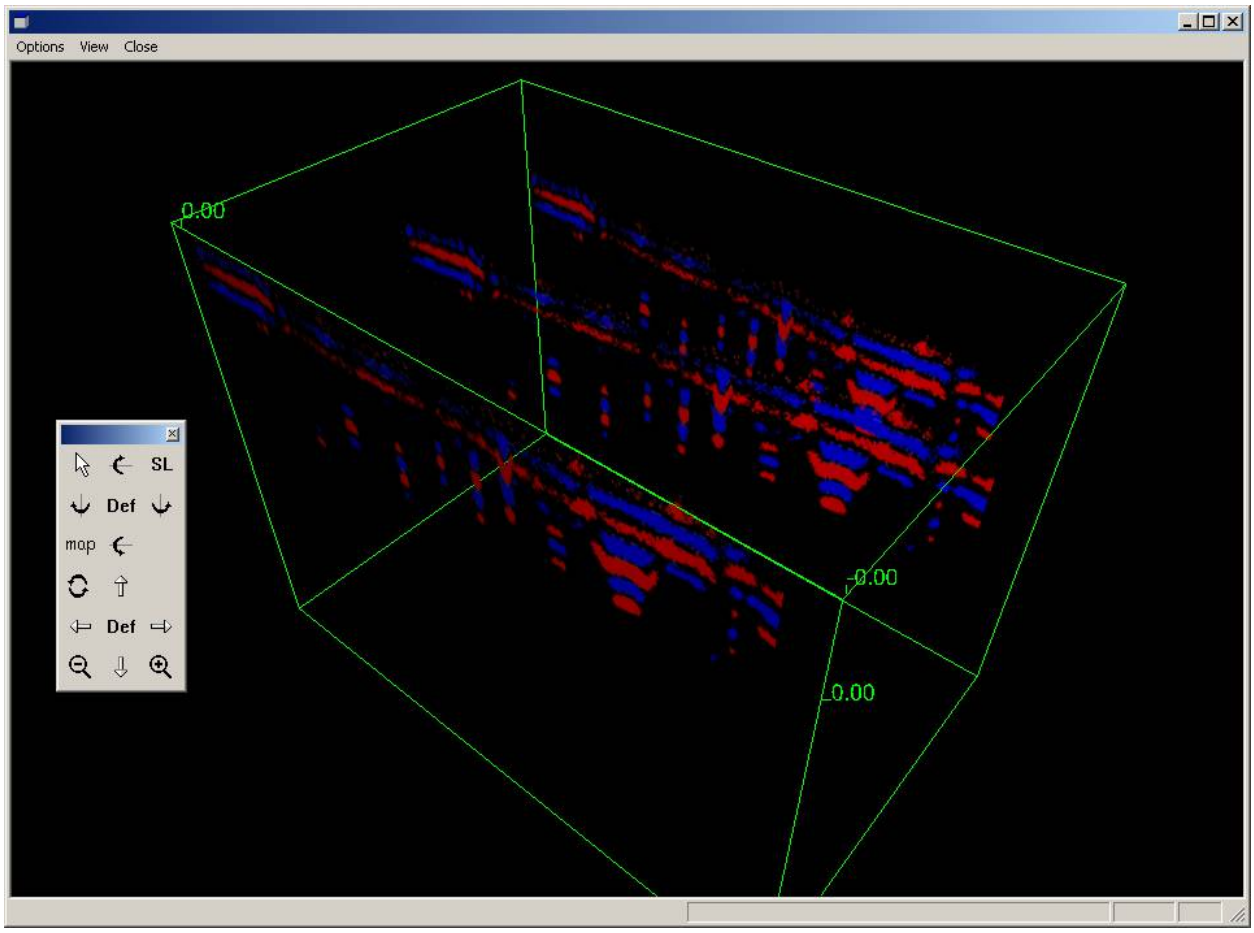


Fig. 9. Data 3D display with transparency – only the high-amplitude portions of the sections are visible.

Axes adjustment

The axes parameters can be adjusted through the dialog window that is called by **Options/Axis** menu command (Fig. 10).

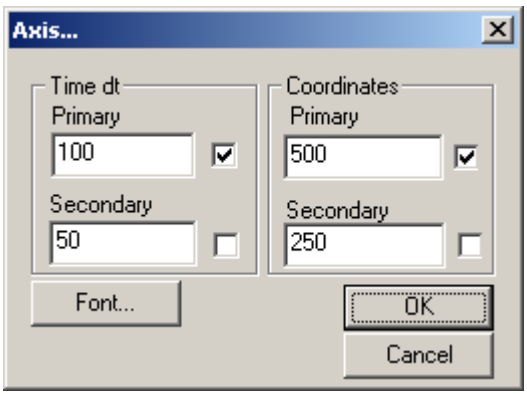


Fig. 10. Axes parameter dialog.

The **Time** group allows adjusting vertical (two-way time) axis. The **Coordinates** group controls X and Y horizontal coordinate axis. For both groups, you can specify **Primary** and **Secondary** tick intervals (for time axis – in nanoseconds, for coordinate axes – in meters). To the right of every field you can indicate if the particular type of ticks is to be displayed on the axes.

Use **Font** button to display a standard dialog for adjusting font to be used for axes tick marks.

Working with the map

The *3D Gazer* allows displaying a map above the seismic sections. A map here is an arbitrary BMP image, which corners are tied to coordinates (see Fig. 4). It could be a survey map with the lines, a chart of a building where the survey was performed, area map with topography etc. The map can be loaded through a dialog window that is called by **Options/Map…** menu command (Fig.11).

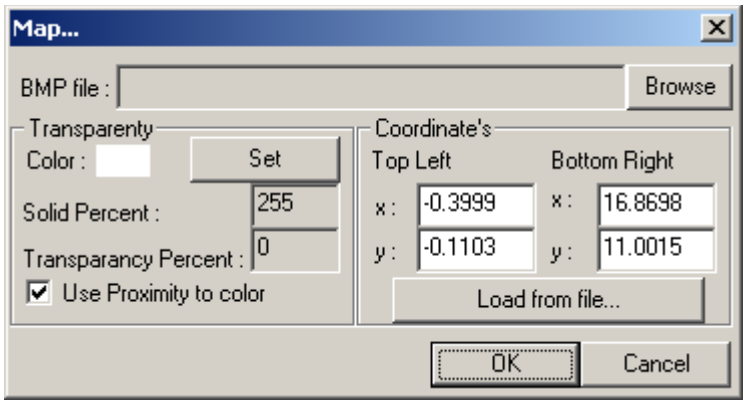


Fig.11. Map parameter dialog.

Use **Browse** button to select a BMP image-file with the map you want to display.

IMPORTANT: An image must be in Window BMP format in 24-bit RGB color mode. In case your map is in a different format, use any third-party image editing software to convert it to the appropriate format.

When the file is selected, its name will be displayed in **BMP file** string.

The **Transparency** group of fields allows adjusting map transparency parameters.

The transparency of the map can be calculated by two methods. If **Use proximity to color** option is on, transparency is calculated automatically for every color of the map through the proximity of this color to the specified opaque color. Use **Set** button to select a color that is to be *opaque*. The other colors of the map will be as transparent as they differ from the specified color.

When **Use proximity to color** option if off, use **Set** button to select the color that is to be *transparent*. All other colors of the map, in this case, will be considered to be *solid*. Then you can separately adjust the actual transparency of the *transparent color* and *solid colors* by means of **Solid Percent** and **Transparency Percent** values. These fields can take on values from 0 (fully transparent) to 255 (fully opaque).

In order to properly tie the map to the data, in **Coordinates** group of fields specify coordinates of **Top Left** and **Bottom Right** corners of the map (in meters). The coordinates can also be loaded from a text file: the 2nd line must contain X and Y coordinates of the top left corner, the 4th line – those of the bottom right corner (see Fig. 12).

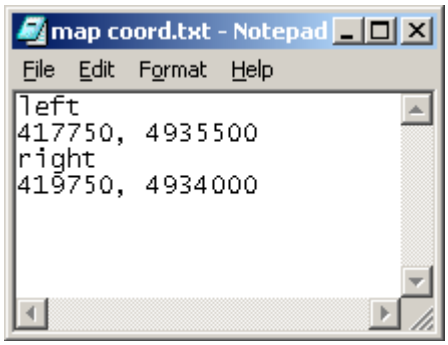
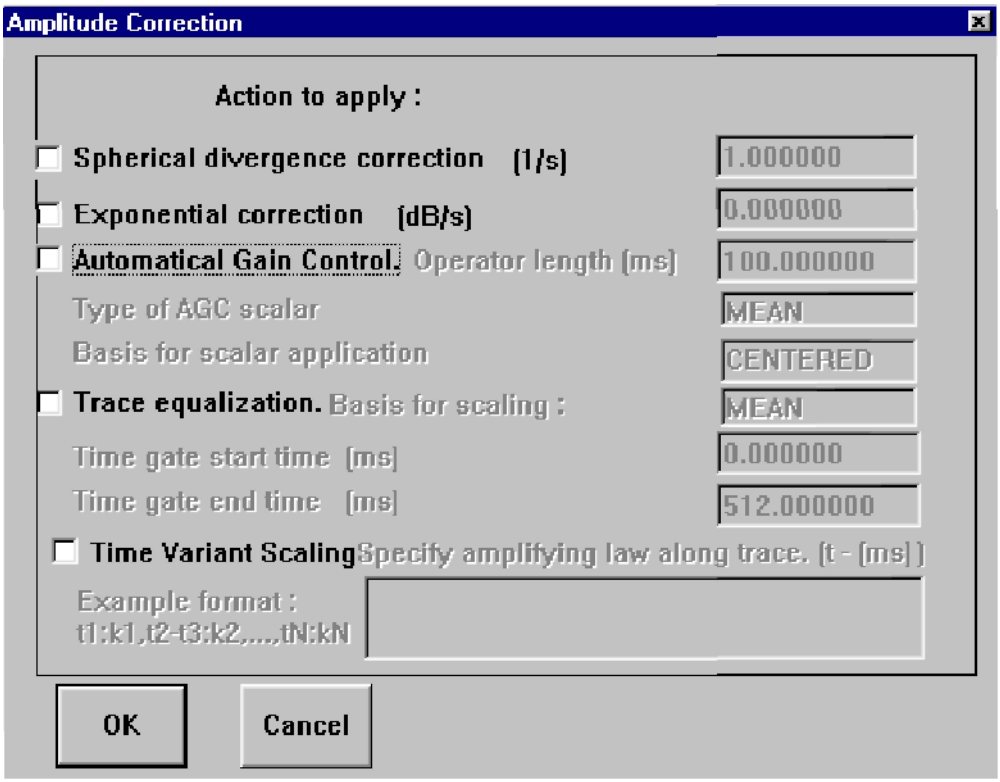


Fig. 12. An example of text file with map coordinate opened in a standard Windows Notepad application.

After the map is loaded, use "Map" button of the Tool Bar to make it visible on the screen.

Amplitude Correction

This module allows for different types of amplitude correction of data. When accessing this module the following window appears:



The options available in this window are described below:

- **Spherical divergence correction:** applies gain function variable in time to traces. Each sample of the trace is multiplied by the corresponding time and by a constant coefficient set by user. Allows compensation for amplitude loss during spherical divergence of wavefront. When activating this option, specify (in l/s) the coefficient for linear correction of gain.

- **Exponential correction:** multiplies every sample by the exponential function of e^{kt}. This allows approximate compensation for intrinsic attenuation in the media. Neither attenuation coefficient variation with depth, or frequency dependency, are taken into account. When activating this option, specify the constant in dB/s to enable the amplitude correction calculation.

- **Automatic Gain Control:** allows automatic changing of gain, applied to trace samples, as a

function of amplitudes in AGC sliding window. For each trace, the gain scalar is calculated for every position of a window of given length, sliding down the trace with the step of one sample. Then, the scalar is applied to the specified sample within the window, i.e. the amplitude of the sample is divided by the scalar. As a result, amplitude variations along the trace reduce. When activating this option, specify:

- **Operator length** - the length (in ms) of the sliding window that will be used for the gain scalar calculation.

- **Type of AGC scalar** - **Mean** is used for AGC scalar calculation as mean of absolute values of amplitudes in the sliding window. **RMS** (root-mean-square) is used for AGC scalar calculation as root-mean-square of the amplitudes within the window. To select a type of AGC scalar, click the left mouse button (MB1) in the corresponding field.

- **Basis for scalar application - CENTERED** applies the scalar to the central sample at every sliding window position **TRAILING** applies the scalar to the last sample in every sliding window. **LEADING** applies the scalar to the leading sample in every sliding window. To chose the position of scalar application, click the left mouse button (MB1) in a corresponding field.

- **Trace equalization** calculates an individual gain scalar for each trace. The scalar is calculated over the indicated fixed time window. Then, all amplitudes of the trace are divided by this scalar. As a result, the amplitude variation between the traces are reduced.When activating this option, specify:

- **Basis for scaling.** Defines the amplitude to be used as a gain scalar: **Mean, RMS, Maximum** (i.e. mean, root-mean-square, or maximum amplitude within the specified window select the basis for scaling, click the left mouse button (**MB1**) in a corresponding field.

- **Time gate start time.** Defines the start time scalar calculation window.

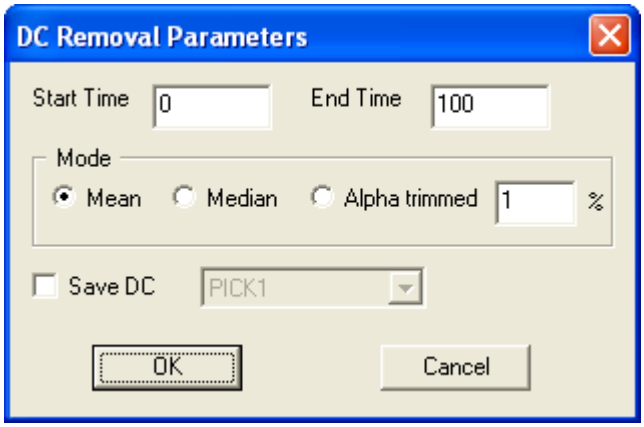
- **Time gate end time.** Defines the end time scalar calculation window.

- **Time variant scaling** defines the law of amplitude scaling along the trace. When the option is activated, specify time-scaling pairs as a text string. The syntaxes of the string is as following:

time1:gain1,time2-time3: gain2,...,timeN:gainN

DC Removal

This module removes DC component of each trace if there is any. When accessing this module the following window appears:



- **Start Time** and **End Time** define the time range where the DC level will be evaluated.

- **Mode** defines the mode of DC evaluation:

Mean - the mean value is taken within specified time range.

Median - the median value is taken within specified time range.

Alpha trimmed - the mean value is taken within specified time range excluding specified percentage of the smallest and the biggest values.

- **Save DC** check box, when checked, allows saving the DC value evaluated for each trace into a trace header field selected in the drop-down list to the right of the check box.

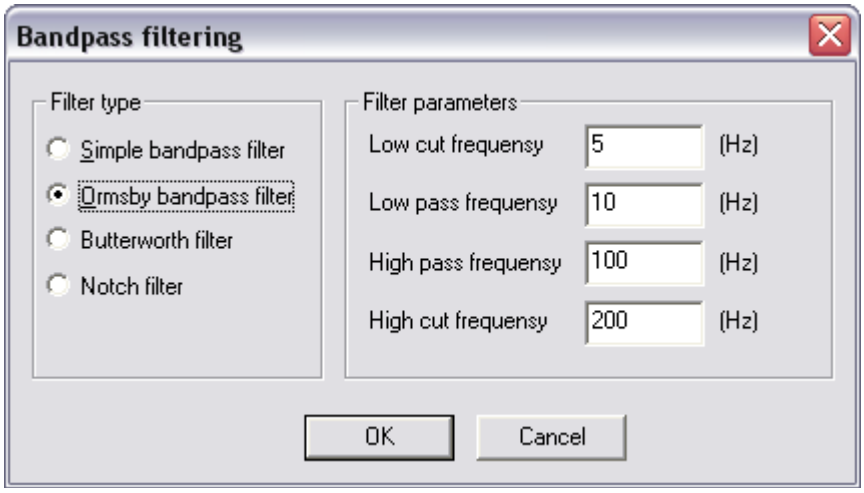
Bandpass Filtering

This module applies frequency filtering to every input trace. The filtering algorithm operates in

frequency domain and is realized by means of multiplying trace Fourier transform on digital

filter frequency characteristic.

When this module is activated the window containing two fields: for filter type selection and for filter parameters selection will appear. The set of filter parameters depends on the type of filter selected.



The field for filter selection contains:

- **Simple Bandpass filter** - simple trapeziform bandpass filter. Here, specify four values of frequency in Hz in filter parameters selection field.

- **Ormsby Bandpass filter.** Here, specify four values of frequency in Hz in filter parameters selection field.

- **Batterworth filter.** Here, specify two values of frequency limits values in Hz and two slope values in dB/Oct.

- **Notch filter** - notch filter is trapeziform. Here, also specify four values of frequencies in Hz.

Choose one type of filter. In the filter parameters selection field for **simple bandpass filter** or for **Ormsby bandpass filter** select a set of four frequencies in the corresponding parameters' fields. These frequencies consistently define the 0% and 100% points of signal gating from the side of lower frequencies and 100% and 0% points of signal gating from the side of higher frequencies (expressed in Hz). The filter slopes are developed within a frequency domain by linear weight

function (for simple bandpass filter) or by Hanning function (cosine weight function) for Ormsby bandpass filter. *Example:*

Low Cut Frequency 5

Low Pass Frequency 20

High Pass Frequency 50

High Cut Frequency 80

If you set such parameters it will result in creation of a band pass filter with a band pass from 20 to 50 Hz and with low frequency slope 30 Hz wide.

For **Batterworth filter**, specify a set of parameters: frequency-steepness-frequency-steepness in corresponding fields. *Example:*

Low Pass Frequency 20

Low Cut Slope 10

High Pass Frequency 50

High Cut Slope 40

If you set such parameters it will result in creation of a bandpass filter with a band pass from 20 to 50 Hz and with 10 dB/Oct slope steepness for lower frequencies and 40 dB/Oct slope steepness for upper frequencies.

For **notch filter** the frequencies consistently define 100% and 0% points of signal gating from the side of lower frequencies and 0% and 100% points of signal gating from the side of higher frequencies. *Example:*

Low Pass Frequency 40

Low Cut Frequency 45

High Cut Frequency 55

High Pass Frequency 60

If you set such parameters it will result in creation of a bandpass filter with a suppression band from 45 to 55 Hz and 5 Hz wide slope for lower frequencies and 5 Hz wide slope for upper frequencies.

Butterworth Filtering

This module applies Butterworth frequency filtering to each input trace. The filtering algorithm operates in the frequency domain and is implemented by multiplying a Fourier transform of the trace by the digital filter frequency response.

Theory

The Butterworth filter is designed in such a way as make its amplitude frequency response as smooth as possible for frequencies within the pass band, but have it drop almost to zero for frequencies within the stop band range.

The amplitude frequency response G(w) of an n^{th} order Butterworth low-pass filter can be expressed by the following formula:

$$G^2(w) = \frac{1}{1 + (\frac{w}{w_c})^{2n}}$$

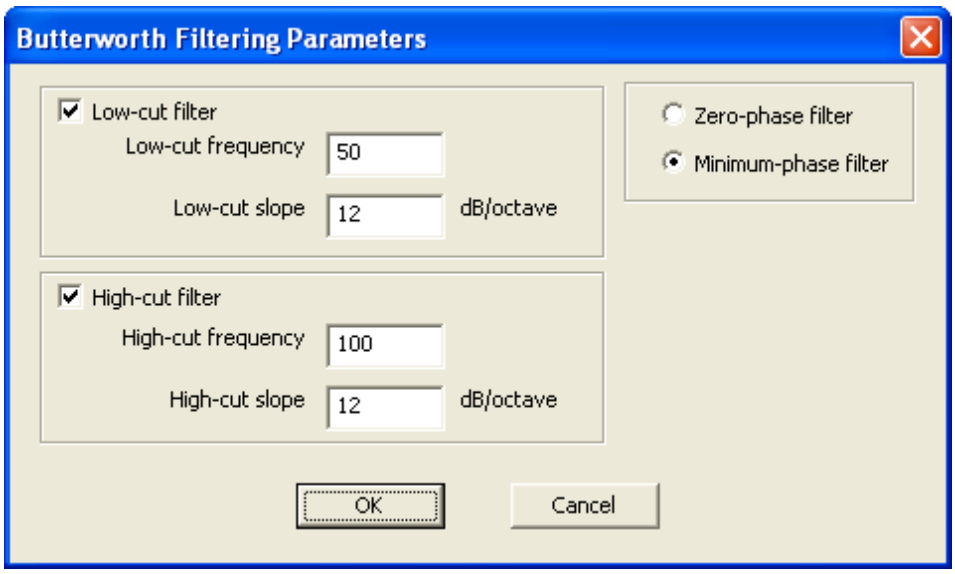
where

- n – filter order (for a first-order filter, the response rolls off at −6 dB per octave, for a second-order filter, the response decreases at −12 dB per octave, a third-order at −18 dB, and so on.)

- w_c – cut-off frequency (frequency on which the amplitude is equal to -3 dB)

The amplitude-frequency response of an n^{th} order Butterworth high-pass filter is calculated using the same formula, but with "w" replaced by "1/w".

Module parameters



When the module is activated, a dialog box with filter parameters will appear:

The dialog box allows specifying the filter operator type, two frequency limit values in Hz, and two filter slope ratio values in dB/oct.

Select the phase of the filter operator:

- **Zero-phase filter –** zero-phase form of the filter operator;
- **Minimum-phase filter –** minimum-phase form of the filter operator;

Specify the type(s) of filter to be applied, the cutoff frequencies and slopes (in dB/oct)

- **Low-cut filter**

Low-cut frequency – the -3dB cutoff frequency;

Low-cut slope in dB per octave;

- **High-cut filter** – low frequency range filtration (high frequency suppression)

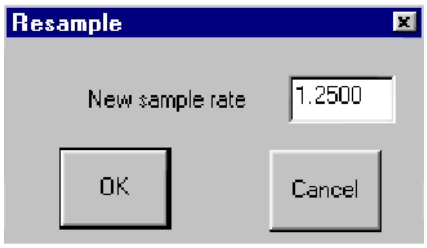
- **High-cut frequency –** the -3dB cutoff frequency;

- **High-cut slope** in dB per octave.

Resample

This module allows for changing of sample rate of the data. It allows both increasing and decreasing of the sample rate. New sample rate can be a broken number. Usually the data are resampled to rarer sampling interval in order to increase processing speed and reduce space on disk required for data storing.

When this module is activated the following window appears:



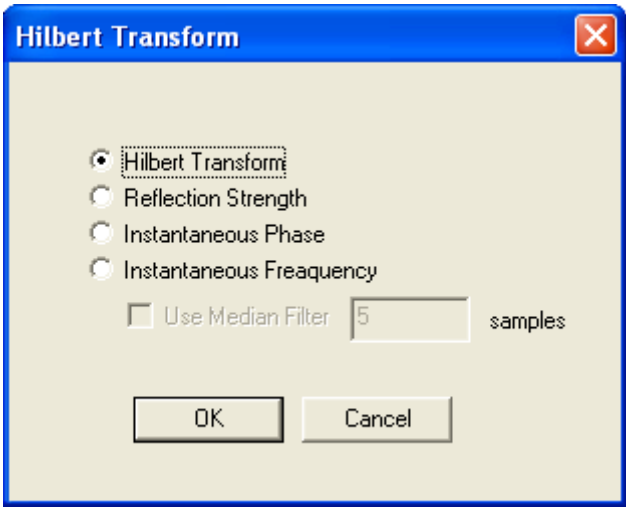
In the **New sample rate** field, specify new sampling interval value expressed in ms.

Hilbert Transforms

This module is used to recalculate seismic traces into traces of reflection strength, instantaneous phase or instantaneous frequency.

For every trace of the dataset through passing the flow its own analytical function (complex trace) is created, where the real part of the signal is the trace itself and imaginary part is its Hilbert transform. The module of this function is usually called instantaneous amplitude or "reflection strength". Reflection strength analysis often allows more accurate tracing of amplitude variations along reflecting boundaries and the whole section. Since a function phase is independent from module size/value, the instantaneous phase study can be used while boundary tracing, faults and irregularities educing. Instantaneous frequency is calculated as phase derivative and can be used while studying the absorbing or/and scattering properties of the section, since absorption and scattering result in frequency-dependent elastic wave attenuation.

When this module is activated the following window appears:

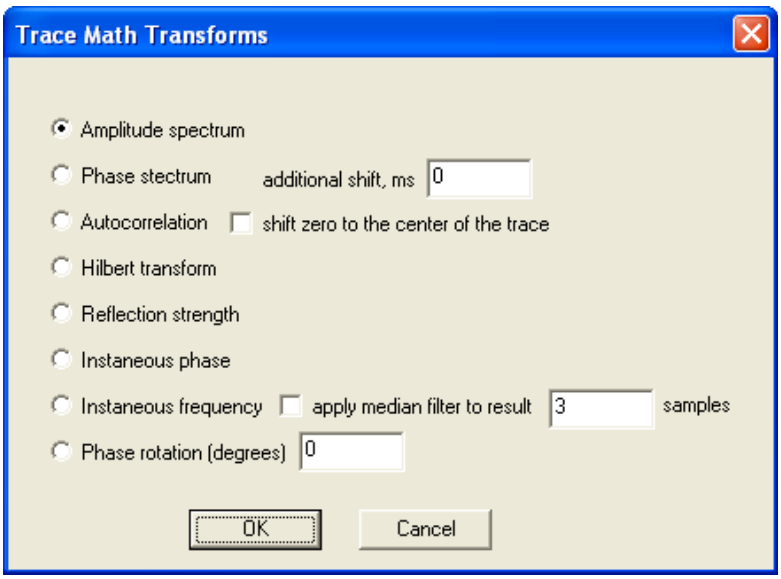


Select one of available types of transform:

- Hilbert Transform
- Reflection Strength
- Instantaneous Phase
- Instantaneous Frequency
- **Use median filter___samples** - in case when instantaneous frequency is selected, you can activate this option to apply median filter (with the operator of the specified number of samples) in order to smoothen the result.

Trace Math Transforms

This module allows some integral transformations of every trace of the dataset through-passing the flow. When this module is activated the following window appears:



Here, select the type of transform:

- **Amplitude spectrum**

- **Phase spectrum.** When this option is selected it is possible to set **additional shift, ms/**

- **Autocorrelation**. By default, time-zero of the autocorrelation function remains at the beginning of the trace. Use the check box to the right to shift zero to the center of the trace.

- **Hilbert transform**;

- **Reflection Strength –** amplitude envelope as calculated through Hilbert transform.

- **Instantaneous Phase** - instantaneous phase (calculated through Hilbert transform)

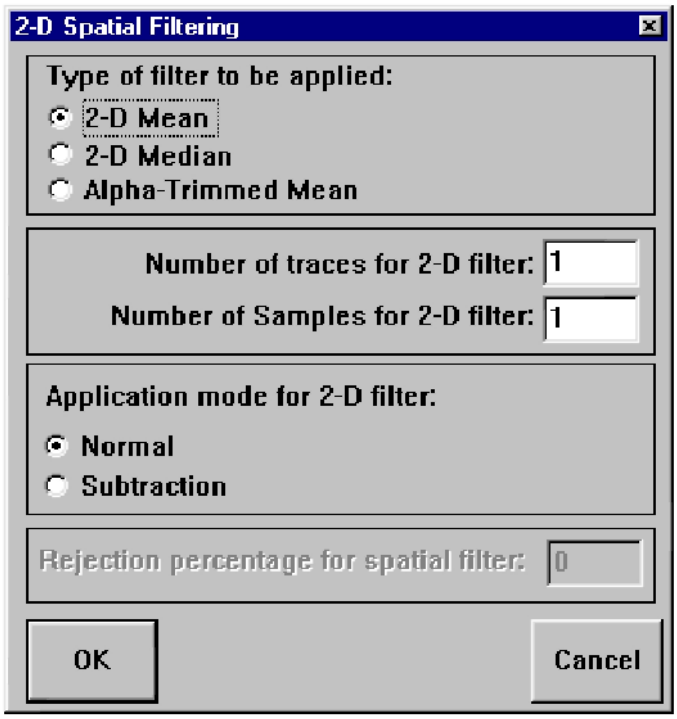
- **Instantaneous Frequency** - instantaneous frequency (calculated through Hilbert transform). When this kind of transform is chosen, the **apply median filter to result** option becomes available. You can switch it on in order to smoothen frequencies via the median filter. The length of the filter operator should be specified in the number of **samples**.

- **Phase rotation** (degrees). When the option is chosen specify the phase rotation in degrees in the edit field to the right.

2D Spatial Filtering

This module is meant to accomplish various types of two-dimensional spatial filtering.

When this module is activated the following window appears:



In the **Type of filter to be applied** field, select the type of filter:

- **2-D Mean** - this algorithm averages the samples within the bounds of filter application window;

- **2-D Median** - this algorithm sorts the samples within the bounds of filter application window and generates median (central) sample of the sorted set.

- **Alpha-Trimmed Mean** - this algorithm sorts the samples within the bounds of filter application window and averages the range of values centralized regarding to the median. This process is equivalent to the process of culling of samples which are beyond the bounds and smoothing the rest of them.

Next, specify the size of window for filter application, i.e. its operator horizontal and vertical

dimensions.

- **Number of traces for 2-D filter** - number of traces to be used as the width of the operator of two-dimensional spatial filter. This value must be an odd integer.

- **Number of samples for 2-D filter** - number of samples to be used as height of the operator of two-dimensional spatial filter. This value must be an odd integer.

There are two modes of filter application. The first mode (normal) substitutes the sample value in the center of operator of two-dimensional filter for calculated value. In the second mode in the center of operator of two-dimensional filter calculated value is subtracted from the input value thus preserving the "differed" dataset.

In the **Application mode for 2-D filter** field, specify the spatial filter mode:

- **Normal** - allows replacing of the initial value in the center of the two-dimensional filter operator by calculated value.

- **Subtraction** - allows subtracting the calculated value from the input value in the center of the two-dimensional filter operator. It is usually applied for culling of energy of some coherent waves.

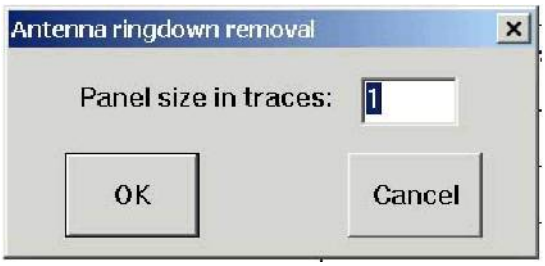
- **Rejection percentage for spatial filter** (this option is available only when **Alpha- Trimmed Mean** filter is selected) - in this field, specify the percentage of input samples of the window of two-dimensional special **alpha-trimmed mean** filter which will be discarded before averaging the rest of the samples. The 40% value means that the lowest 20% of the samples and the highest 20% of the samples will be discarded while filter calculation. When the 0% value is entered, the filter at the output will give the mean sum whereas the 100% value will give the median filter.

WARNING!: Owing to the median filters nature, when using the median filter the accidental gaps in values of samples may occur and they are the result of shifts in family of samples taken place while process of sorting. It is recommended that the median filter is followed by the bandpass filter with a wide bandpass in order to eliminate these calculation gaps.

Antenna Ringdown Removal

It is obvious that the presence of constant component in all the traces (or in part of traces) of the window under processing can be only an instrumental error. It can be removed by means of subtracting the mean for the window from every sample of all traces of the ensemble. Practicably it is possible that the value of this constant component slowly changes within the window under processing. A special parameter for window size is available in the module which equals average number of traces. ARR can be considered as the most rapid and simplified mode of *2D* spatial filtering.

When this module is activated the following window appears:



Panel Size - defines number of traces which will be averaged in order to find the constant component.

Radial Trace Transform

Theory

The module is designed to transform traces from (x,t) domain into (v,t') domain and back following the rule (see Fig. 1):

$R(S(x,t)) = S'(v,t)$

$R^{-1}\{S'(x,t')\}=S(x,t)$

$t'=t;\ v=x/t$

where x – distance from source to receiver, t – time, v – apparent velocity.

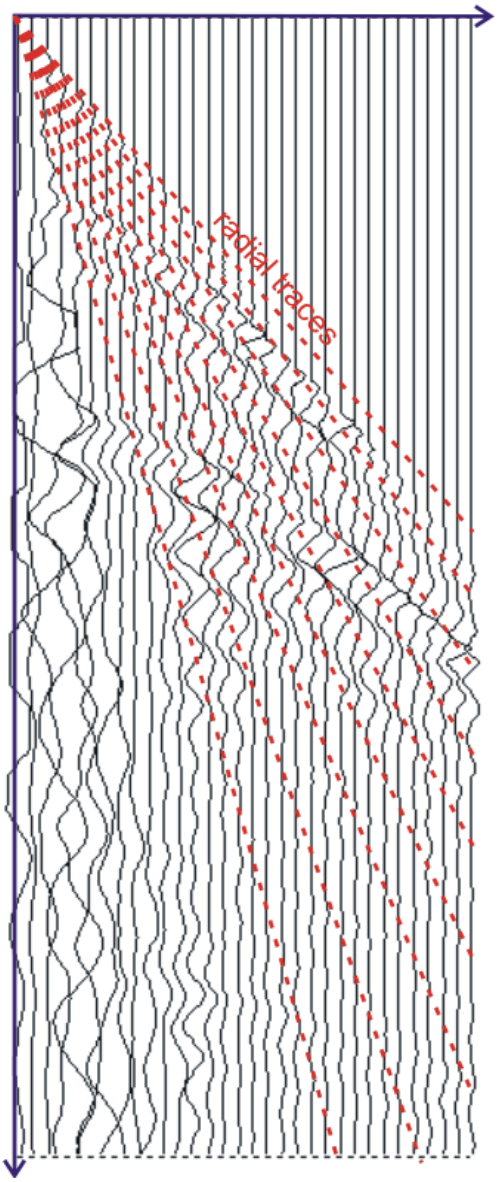


Fig. 1. Transformation into radial traces

Several filtering types (special and time) and different muting types can be applied to traces in V-T domain.

The transform can be used for noise attenuation, that regards coherent noise with linear seismic events, starting from the axes' origin (for instance, direct and air waves etc.).

Parameters

The figure (Fig. 2) demonstrates a dialog box for parameters' setting of the module Radial Trace Transform.

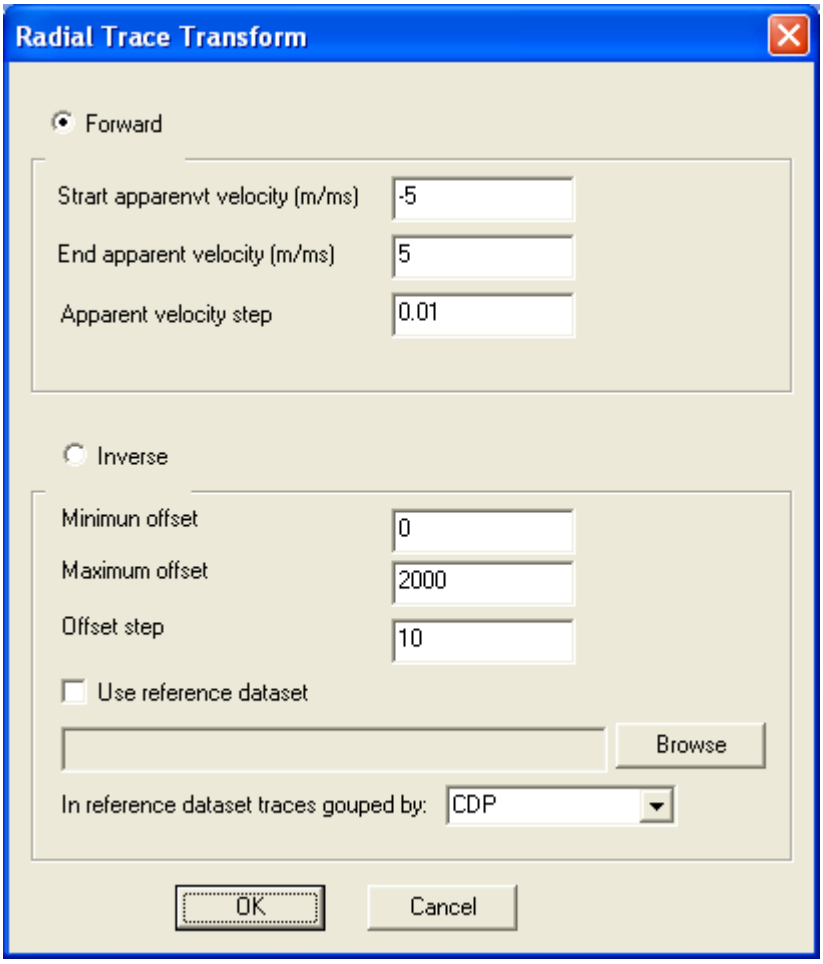


Fig. 2. Parameters of module Radial Trace Transform

- Forward transform (**Forward**).

- **Start apparent velocity** – minimum apparent velocity (in m/ms or km/s, which is similar)

- **End apparent velocity** – maximum apparent velocity (in m/ms)

- **Apparent velocity step** – step of search (step between the generated traces, in m/ms)

- Inverse transform (**Inverse**)

Inverse transform can be performed in two modes – into the set of offsets specified by the user (in this mode a trace ensemble in x-t domain with the specified set of offsets will be obtained from each radial traces' ensemble). The following parameters are applicable for this mode

- **Minimum offset** (in m)

- **Maximum offset** (in m)

- **Offset step** (in m)

The second mode of inverse transform is selected via the check-box **Use reference dataset**. In this case the module generates traces with the same geometry (offsets) and header fields, as in the seismic dataset from the database, selected by the user. The selection of the database object is performed by standard means after clicking the button **Browse**.

It is assumed that the offsets are kept within the header field OFFSET of the raw data when performing the forward transform, The traces are formed as a result of transform, with the header field OFFSET containing apparent velocity. The other way round is done during the inverse transform.

IMPORTANT! The traces' sorting in Reference dataset should be as follows: *ENSEMBLE_INDICATOR*:OFFSET, where *ENSEMBLE_INDICATOR* – header field, specified in **In reference dataset traces grouped by…** parameter, the same as the first sorting field of input radial traces.

IMPORTANT! Each ensemble of input traces in any domain (x-t or r-t) should contain at least 2 traces.

Application

Let's give a typical example of module application aimed at ground roll suppression. Let us assume that the raw data have a header field TRC_TYPE = 1.

- **Forward transform**

- **Trace Input (FFID:OFFSET sorting) <- Dataset A**

- **Radial Trace Transforms**

- **Forward**
- **Apparent velocity -2:0.01:2 (км/с)**
- **Trace Output → Dataset B**

- **Generation of a reference dataset**

- **Trace Input (FFID:OFFSET sorting) <- Dataset A**
- **Trace Output → Dataset C**

- **Generation of ground roll model and inverse transform**

- **Trace Input (FFID:OFFSET sorting, range OFFSET -0.35-0.35) <- Dataset B** *(OFFSET range (where the apparent velocities are kept at the moment) is limited by apparent velocities of ground roll)*
- **Bandpass Filter** - 0-0-8-12 (Hz) (pass only the ground roll)
- **Radial Trace Transforms**
- **Inverse**
- **Use reference dataset** (Dataset C, group header FFID)
- **Trace Header Math**
- **TRC_TYPE = 2**
- **Trace Output → Dataset D**

In this example, to distinguish the ground roll a band pass filtering (high cut filter) is used. However instead of the band pass you may use some other 1D or 2D filters to pickup the field of noise in R-T

domain.

- **Subtracting ground roll model from the raw data**

- **Trace Input <- Dataset A, Dataset D** *Parameters should control the traces sorting in pairs in the flow: "minuend1", "subtract1", "minuend2", "subtract2"...*

- **Trace Math**

- **Trace Output → Dataset E**

This is a preferential kind of approach (when the noise model is generated, and subtracted from the raw data), in comparison with the direct approach (when the noise is subtracted in R-T domain), because an inverse transform can (though moderately) distort the data. Moreover, it allows editing the noise model before subtraction.

Reference

Henley, D.C., 2003, Coherent noise attenuation in the radial trace domain. Geophysics, 68, p. 1408.

Radon Transforms

Theory

Forward linear discrete Radon transform of seismogram, containing N traces $L^* L$ with a range of offsets h from L_0 to L_1 can be specified as

$$v(p,\tau) = (Lu)(p,\tau) = \sum_{l=L_0}^{L_1} u(h_l, \tau + h_l p)\Delta h_l$$

Correspondingly the discrete version of inverse Radon transform is

$$\tilde{u}(h,t) = (L^* v)(\tau, p) = \sum_{j=J_0}^{J_1} v(h, t - hp_j)\Delta p_j$$

Where $\Delta p_j = p_{j+1} - p_j$ denominates an increment of ray parameter. The Fourier transform of the given equations on time variable gives

$$V(p,f) = \sum_{l=L_0}^{L_1} U(h_l,f)\exp(2\pi\, ifh_l p)\Delta h_l$$

$$\tilde{U}(h,f) = \sum_{j=J_0}^{J_1} V(p,f)\exp(-2\pi\, ifhp_j)\Delta p_j$$

In matrix notation (asterisk signifies operator contingency), the data is a result of inverse Radon transform application $d = Lm$, then (Yilmaz 1994), we need to find m while minimizing the following functional of a square norm

$$\| d - Lm \|^2$$

it corresponds to a well-known formula of the least square method

$$m = (L^* L)^{-1} L^* d$$

As the matrix $L^* L$ is very close to degenerate matrix, it is worth using a regularizing parameter μ, that modifies diagonal line:

$$m = (L^* L + \mu I)^{-1} L^* d$$

The realization of forward and inverse parabolic Radon transform is similar on the whole.

The transform can be used in order to eliminate different coherent noise with linear event (for example, direct and air waves etc.).

Parameters

Figure (Fig. 2) illustrates a dialog window for parameters' setting of Radon Transforms package.

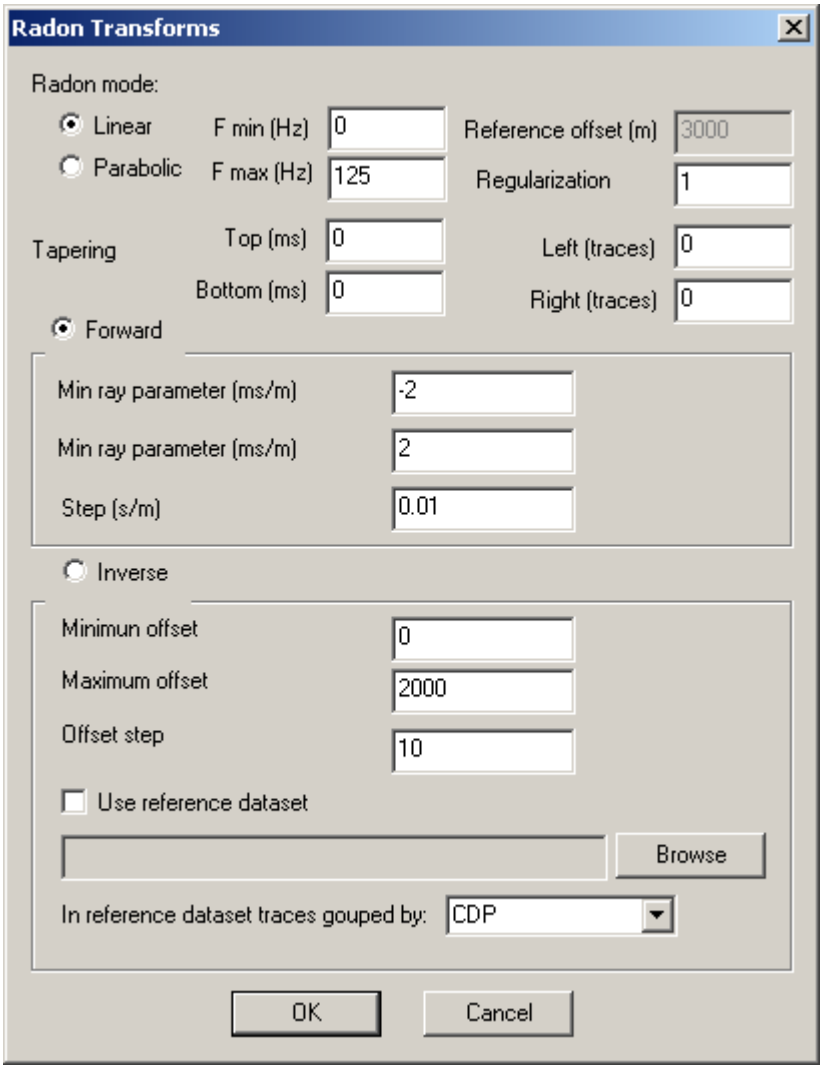


Fig. 3. Parameters of the module Radial Trace Transform

- **Radon mode** – Radon transform mode (Linear or Parabolic)

- **Fmin / Fmax** – frequency range (Hz) of the signal, in data subject to transform.

- **Reference offset** – offset (m), for which a range of moveouts will be indicated in case you choose Parabolic mode. As a rule, it should be approximately equal to the maximum offset in seismograms. Irrelevant in case of the linear Radon transform mode.

- **Tapering** – ensemble edges' smoothing (in space (Left, Right, traces (number)), as well as in time (Top, Bottom, ms)). With the aim of distortion reduction, linked with the calculation of discrete Fourier transform, the ensemble is multiplied by piecewise-cosine weight function.

- Forward transform (**Forward**).

- **Min ray parameter** – minimum ray parameter (in ms/m or s/km, for linear transform and ms for parabolic transform)

- **Max ray parameter** – maximum ray parameter (in ms/m or s/km, for linear transform and ms for parabolic transform)

- **Step** – search step (step between generated traces, in ms/m)

- Inverse transform (**Inverse**)

The inverse transform can be carried out in two modes – into the set of offsets specified by user (in this case each ensemble of radial traces produces trace ensemble in x-t domain with the specified set of offsets). The following parameters are applicable in this mode:

- **Minimum offset** (in m)

- **Maximum offset** (in m)

- **Offset step** (in m)

The second mode of the inverse transform is carried out using the check-box- **Use reference dataset**. In this case the traces with the same geometry (offsets) and the header fields as in the set of seismic data, selected by user, are generated by the module. The selection of database object is performed by standard means after you click the button **Browse**.

It is assumed in the module, that in the raw data the offsets are kept in header field OFFSET when you perform forward transform, as a result of transform the traces are formed with the header field OFFSET, containing ray parameter (or corresponding moveout on a reference offset). The inverse transform is done vice-versa.

IMPORTANT! Trace sorting in Reference dataset should be the following: *INDICATOR_ENSEMBLE*:OFFSET, where *INDICATOR_ENSEMBLE* – is the header field, set in

parameters field **In reference dataset traces grouped by**…, the same as the sorting field, of the input radial traces.

IMPORTANT! Each ensemble of input traces in any domain (x-t or r-t) should contain at least 2 traces.

Application

The application of module is generally the same as the **Radial Trace Transform** module and is laid down in the corresponding section of this Manual.

Burst Noise Removal

The module is designed for removal of high-amplitude noise bursts from the seismic traces.

The algorithm of the module operation is as following:

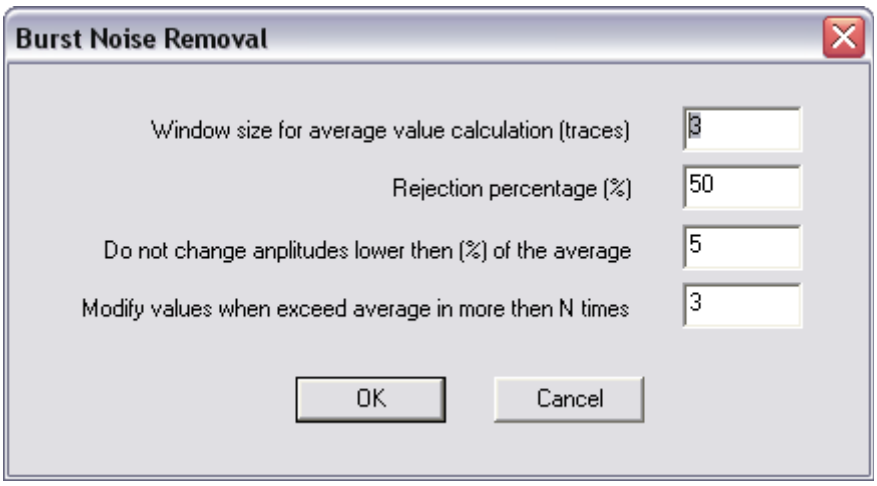
- The average of the absolute amplitudes over all samples of all traces within the flow is calculated.

- Within a sliding window of several traces, alpha-trimmed average of absolute amplitudes is calculated for each sample.

- The average for the current sample within the window is compared with the average over the flow. If the average for the sample over the window is too small (that is, does not exceed certain percentage of the average over the flow), this sample is skipped as the module is not supposed to change low amplitude samples.

- Otherwise, the amplitude of this sample at the middle trace of the window is compared with the average for this sample over the window. If the absolute amplitude exceed the average N times and more, this sample at the middle trace is considered to be a burst and its amplitude is substituted by the alpha-trimmed average for this sample over all traces within the current position of the window.

The parameter dialog of the module looks as following:



Here:

Window size for average value calculation (traces) — number of traces of the sliding window used for alpha-trimmed average calculation.

Rejection percentage (%) for alpha-trimmed average calculation. This percentage of the highest and lowest amplitudes will be rejected, the remaining will be averaged.

Do not change amplitudes lower than (%) of the average — this is the threshold percent from the average over the flow. When the alpha-trimmed average over the window is below this threshold, this sample is skipped and guaranteed to remain unchanged.

Modify values when exceed average in more than N times — set the N-parameter here. When the absolute amplitude of the sample at the middle trace is N-times higher than the average for this sample over the windows, it is substituted by the average.

Wave field subtraction

Theory

This module performs adaptive subtraction of one wave field from another. The main feature of the algorithm is its capability to make adjustments for possible gradual amplitude distortion along the time coordinate. A shaping filter is calculated using the least square method in such a way as to make the filtered subtracted field as close to the source field in terms of minimum quadratic deviation as possible. This filter is essentially a multi-channel filter. Additional channels for its

calculation and subsequent folding are taken from the source trace by adding parametrized time non-stationarity to it.

Method theory

The adaptive wave field subtraction method consists in finding a continuous time shaping Wiener filter $\hat{g}(t)$: in the set of Wiener filters $\tilde{g}(t)$ to satisfy the following condition:

$$\hat{g}(t) = \arg\min_{\tilde{g}(t)} \int_t (p(a,b,t) - \tilde{g}(t) * l(a,b,t))^2 dt \,, \quad (1)$$

where $p(a,b,t)$ is the source trace of the wave field being reduced, a and b define the spatial coordinates of the trace, and $l(a,b,t)$ is the trace of the wave field being subtracted.

The implemented algorithm allows making adjustments for possible discrepancies between the $p(a,b,\ t)$ and $l(a,b,t)$ traces (including nonstationary t discrepancy), but does not have enough degrees of freedom to make adjustment for arbitrary differences.

A traditional method of accounting for nonstationarity in adaptation tasks is a “local stationary” (i.e. stationary within the window) multi-window filtration that involves breaking the trace down into intervals:

$$\hat{g}_i(t) = \arg\min_{\tilde{g}_i(t)} \int_{\hat{T}_i}^{\hat{T}_i} (p(a,b,t) - \tilde{g}_i(t) * l(a,b,t))^2 dt \,, \quad (2)$$

where i is the window number, with its upper and lower boundaries serving as limits of integration Ti. Drawbacks of this method lie in the very task of making adjustments for nonstationarity by breaking the trace down into intervals within which such effects are considered insignificant. As a result, filters generally have to be set up in small windows; this considerably degrades the statistical properties of evaluation and leads to a potential usable reflection energy decrease during subtraction. Besides, such parametrization of nonstationarity nature is described by uneven filter

changes at window boundaries. By introducing matrix designations for the filter set $\hat{g}$, $\tilde{g}$, expression (1) may be presented as follows:

$$\hat{\mathbf{g}} = \arg\min_{\tilde{\mathbf{g}}} \sum_i \int_0^T (p(a,b,t) - (w_i(t)l(a,b,t)) * \tilde{g}_i(t))^2 dt, \qquad w_i(t) = \begin{cases} 1, t \in (\check{T}_i, \hat{T}_i) \\ 0, t \notin (\check{T}_i, \hat{T}_i) \end{cases} \qquad (3)$$

where T is the trace length and wi (t) is the step function. To eliminate uneven change of filters, the number of the shaping filter's degrees of freedom should be reduced while maintaining its nonstationarity, i.e. the filter should be allowed to make adjustments for possible dynamic variations, but not in an arbitrary mode. Use of smooth functions for subtracted trace weighing is suggested to eliminate the drawbacks inherent to the traditional approach.

In this case a substitution of step functions wi (t) with, for example, polynomials $wi(t) = t^i$ can be used. After that, the optimization task comes down to calculating a nonstationary, gradually variable shaping filter for the entire trace. The procedure is technically implemented as follows. First the subtracted trace $l(a,b,t)$ is represented as a weighted set of smooth functions wi (t), resulting in a set of channels li $(a,b,t) = l(a,b,t)wi$ (t). Than adaptation is carried out by calculating a continuous time shaping filter for each channel li (a,b,t).

To increase reliability of filter calculation (especially for traces with high amount of interference), calculations should be performed taking adjacent traces into account too, – for example, $l(a,b + \Delta x,t)$, $l(a,b - \Delta x,t)$ etc., where Δx is the lateral increment. In this case the task of adapting the subtracted wave field to the source data is reformulated as follows: it is also necessary to find minimizing functionals $\hat{g}_k(i,t)$ и $\hat{g}'_k(i,t)$

$$\hat{\mathbf{g}}, \hat{\mathbf{g}}' = \arg\min_{\tilde{\mathbf{g}}, \tilde{\mathbf{g}}'} \int_0^T \left(p(a,b,t) - \sum_{k=-M}^{M} \sum_{i=0}^{N} \left(\tilde{g}_k(i,t) * l_i(a,b+k\Delta x,t) \right) + \left(\tilde{g}'_k(i,t) * l'_i(a,b+k\Delta x,t) \right) \right)^2 dt \quad (4)$$

which leads to Levinson algorithm for multi-channel filters. The block Toeplitz matrix obtained as a result of minimization can become ill-conditioned, which is characteristic of all tasks of optimum spatial filtering. To avoid this, let us find a separate optimum multi-channel filter by variable i for each trace in the database ($b - M\Delta x, b + M\Delta x$), obtaining the following as the result:

$$\tilde{l}(a,b+k\Delta x,t)=\sum_{i=0}^{N}\left(\hat{g}_k(i,t)*l_i(a,b+k\Delta x,t)+\hat{g}'_k(i,t)*l'_i(a,b+k\Delta x,t)\right). \quad (5)$$

Then let us find coefficients *c^k* satisfying the following conditions:

$$\hat{c}_k=\arg\min_{\hat{c}_k} J_{a,b}(\tilde{c}), \qquad J_{a,b}(\tilde{c})=\int_0^T\left(p(a,b,t)-\sum_{k=-M}^{M}\tilde{c}_k\tilde{l}(a,b+k\Delta x,t)\right)^2 dt. \quad (6)$$

The procedure is repeated several times with $p(a,b,t)$ in the last expression substituted with the subtraction result from the previous iteration until the minimum of functional *Ja,b* () is reached.

Recalculation for a larger sampling interval (or, to be more precise, “band transformation” or *resampling*) presents another reserve to draw upon in order to improve the task conditionality. In this case the lower limit frequency of the source trace operating band will correspond to Fourier spectrum zero frequency, while the upper limit frequency of the band will correspond to Nyquist frequency.

Further improvement of algorithm stability in terms of increasing the regular noise suppression depth while maintaining the signal dynamics can be achieved by using spatial smoothing

$$Y_{a,b}(\tilde{c})=\sum_{n=-A}^{A}J_{a,b-n}(\tilde{c}), \quad (7)$$

i.e. averaging of functional based on (2*A*+1) traces. In this case the optimum adaptive filter is determined by minimizing the functional *Ya,b* (c). If such procedure modification is implemented, the filter will be estimated for a group of traces, resulting in regularized solutions.

Parameters

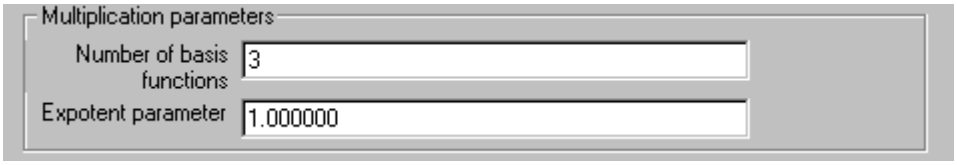
- Input data.

This key assigns the mode of module operation.

As a rule, the raw field is considered to be the minuend field, while the multiples' field, constructed using a tricky approach, is considered to be the subtrahend. The given processing mode is organized to input traces' pairs (**One model dataset** mode), the first trace is related to the raw field, while the second trace of the pair – to the subtrahend field. Thus, the traces of the subtrahend and the minuend fields, sorted by the main sort field are fed to the processing flow (for example, DEPTH) and within this sorting type they are sorted by field, defining the affiliation to the field type (for instance minuend or subtrahend, TRC_TYPE).

The procedure can be used in iterative mode, while subsequently subtracting several fields (for example, several modeled fields of multiples from different boundaries). The program provides (**Two model datasets**) mode when two subtracted fields are fed into it simultaneously. In such a case, both subtrahend fields are considered during the calculation of the shaping field using the least squares technique, which is different from the successive subtraction of each subtrahend field in separate mode. In this mode the traces are input in threes, the sorting is the same as in case of the single subtrahend field.

- Multiplication parameters



The given parameters set describes the characteristics of the basis functions that contribute to multiplication of subtrahend field traces. In particular the procedure algorithm includes the calculation of the multichannel shaping filter. More particularly, the assignment of each multiplication function adds a channel, formed by the multiplication of the raw trace by function

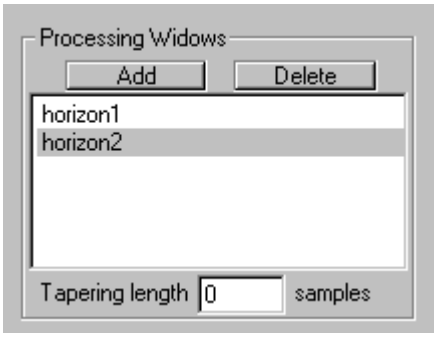
$t^{\alpha \cdot n}$, where

n – number of multiplication function, while alpha – is the so called **Exponent parameter**, used for

fine tuning, and usually is equal to 1. Thus, without consideration of the **Exponent parameter,** we can say that if the **Number of basis function** is 0, the shaping filter is considered to be single channel; if this parameter is equal to 1, the trace, multiplied by time is added to the shaping; if it is equal to 2, a trace with squared non-stationarity is added etc.

The more we assign multiplication functions the better we subtract one field from the other; but we have to take care, as in this case we can subtract undue value from the raw field. You should come to a compromise, and the number of multiplication functions is selected individually proceeding from the processor's experience in each case. You should also keep in mind that the extent of subtraction depends on the other parameters, particularly the filter length and the length of the working window. The more the filter length and the less the window length, the better we subtract (we imply that we, possibly, subtract undue value).

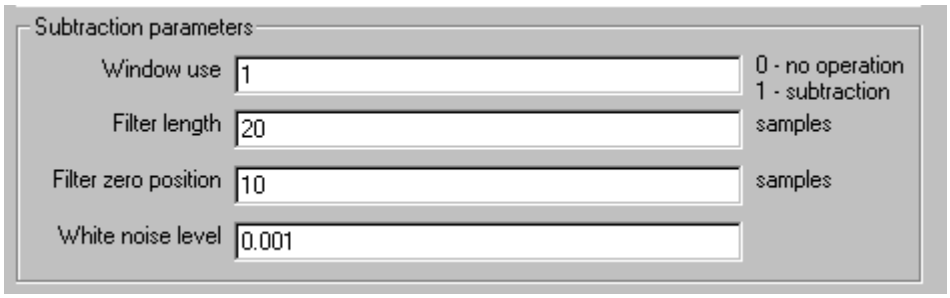
- **Processing windows.**



Considering the nature of the records, sometimes the processing should be done in different windows individually. The procedure supports the division by windows using picks as processing windows' boundaries. Thus if there is no boundary in the list, we conclude that there is only one window from the start to the end of the trace. If only one pick is assigned, the whole processing area is divided into two windows: 1) from the start to the pick, 2) from the pick to the end of the trace. If you assign two picks, we have three processing windows respectively, etc. The boundaries of picks are added/removed using the buttons **Add** and **Delete**, the current set of boundaries is represented in the list. The user should insure that the boundaries do not intersect, since the program behavior is unpredictable over intersecting boundaries. As well you shouldn't use windows that are too narrow, as there will be subtraction to zero which is not good. However, sometimes it is useful to set a narrow window and mark it as non-active (i.e., there will be no subtraction in this window); in such a way you can perform a so-called muting of areas that are unsuitable for subtraction.

Tapering length parameter sets an area of results' stitching in different windows. This is measured in samples.

- Subtraction parameters



The given set of parameters characterizes the parameters of the shaping filter.

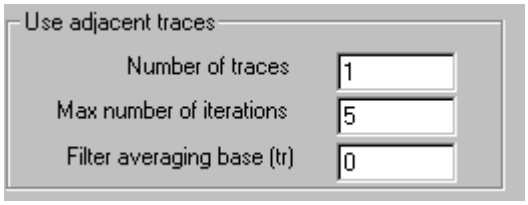
Window use - sets the sign of whether to perform subtraction in the given window or to keep the raw field unchanged. (1 – subtraction in the window should be performed, 0 – should not be performed).

Filter length - sets the length of shaping filter, in samples.

Filter zero position - sets the number of zero filter samples, i.e., determines the ratio of predicted and reminiscent filter components.

White noise level - regularization parameter. Fractional additive to the main diagonal of autocorrelation matrix while solving the system, (for some reason this is sometimes called "white noise level".)

- Use adjacent traces



Number of traces sets additional channel number in spatial coordinate. It is useful to consider not

only the subtracted trace but also adjacent traces while forming the filter and performing subsequent subtraction. This parameter sets the arm of the adjacent traces' set window. That is, when it is equal to zero, the adjacent traces are not used in the calculation; if it is equal to 1, then 3 traces are used: the central trace and each adjacent trace at each side, etc. As adding new channels is linked with some difficulty, another algorithm for reporting adjacent traces is created. Namely, first we calculate the filter for each spatial channel, the subtracted traces are filtered, and then the coefficients for each spatial channel are matched in order to minimize dispersion using a special algorithm (Kholetsky algorithm). Then each filtered channel is subtracted from the raw trace using the corresponding coefficient. In order to increase the method efficiency a notion of iteration is introduced. Namely the given procedure is performed several times. The filter for every following iteration is calculated on the basis of new cross-correlation matrix. A comparison of trace energy difference before and after iteration is performed after every iteration; if it exceeds the calculation accuracy, the cross-correlation matrix is recalculated and transition to the next iteration is performed.

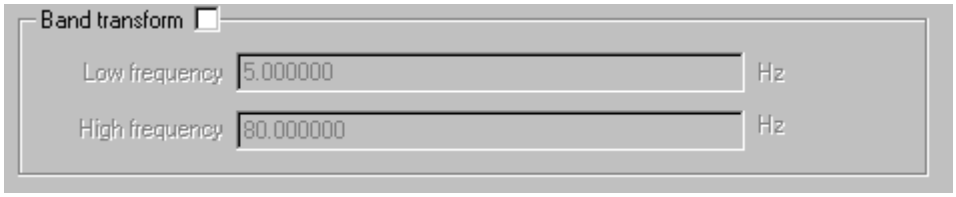
- **Max number of iterations** parameter sets the maximum number of such iterations. After having achieved this number the iteration cycle is automatically stopped.

- **Filter averaging base** – another way of allowing for adjacent traces. It sets the arm of the averaging base of autocorrelation and cross-correlation matrices during filter calculation. In other words, the shaping filter is averaged by a certain number of traces. Let's point out that filter averaging can be done simultaneously with spatial channels.

- Other parameters

- **Accuracy** – accuracy of calculation. Parameter is used while comparing the traces' energy before and after filtering.

- Band transform



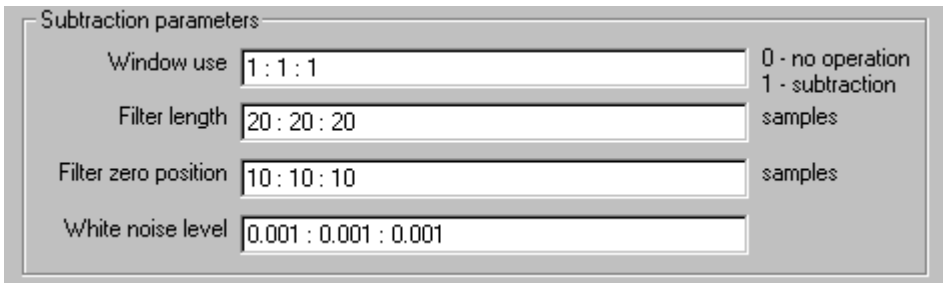
You can perform filtering, within the full bandwidth as well as within a limited bandwidth. The button **Band transform** is designed to toggle between the modes. When you use the mode of limited bandwidth, a kind of re-sampling, or, more precisely, a band transform of the data is carried out before the subtraction in such a way that the specified bandwidth is projected into the whole available bandwidth. . After the subtraction, an inverse band transform is applied to the result. Due to this procedure, the subtraction is performed within the limited frequency band and the remaining frequencies are filtered out. This stratagem is used because the algorithm is designed for the work in full frequency range, while practically speaking it is preferable to use a limited bandwidth. **Low frequency** and **High frequency** parameters assign low and high frequencies of the used bandwidth, correspondingly.

Remarks on parameters assignment in several processing windows

Due to properties with depth variation it is helpful to divide the whole data area into processing windows, and to use specific processing parameters for each separate window. The following parameters can be set individually for each window: **Number of basis function, Exponent parameter, Windows use, Filter length, Filter zero position, White noise level, Low frequency,** and **High frequency.**

In this case the values are specified using a colon ':' as the separation character between the values for individual windows.

For example, subtraction parameters for three processing windows can be written as follows:



F-X Deconvolution

This module is used to suppress random noise both on trace ensembles and stacked sections.

Principle of operation

The F-X deconvolution procedure is based on prediction of linear events in the frequency-space domain.

A linear event in the time domain described by the expression $f(x,t) = \delta (a+bx-t)$ is transformed using a Fourier transform into $f(x,\omega) = e^{i\omega(a+bx)}$ or $f(x,\omega) = e^{i\omega a}(\cos(\omega bx)+i\sin(\omega bx))$ in the frequency domain. For simple linear events this function is a periodic function by x. This periodicity (sinusoidal complex signal along a certain frequency section) can be traced along any constant frequency (frequency section) in the frequency-space (f-x) domain.

The F-X deconvolution procedure uses a complex Wiener filter to predict the signal for one trace

ahead. The Wiener prediction filter is calculated for spatial series obtained at each frequency by means of the Fourier transform. Each prediction filter is sequentially applied in two directions (forward and backward in space), and the results are then averaged to eliminate prediction errors.

Prediction in the f-x domain is applied in small windows to ensure that the assumption about the linearity of events in the time domain is valid.

Module parameters:

- **Filter length** – number of samples in the prediction filter.

- **White noise level** – percentage of white noise added to the zero delay of the auto-correlation value. Default value is 1%. Increasing this value eliminates more noise, making the data "smoother".

- **Horizontal window –** number of traces in the horizontal F-X prediction window. The value must be greater than the number of filter samples.

- **Time Window –** time window length in ms.

- **Time window overlap** – amount of time window overlap. Is added at the top and at the bottom of the time window for "vertical mixing" purposes.

- **Start frequency** – minimum signal frequency in the data.

- **End frequency** – maximum signal frequency.

- **Divide by ensembles** – the F-X deconvolution function will be applied within ensembles. Ensembles are determined based on the Trace Input module headers.

- **Mute hard zeros** – if the data contain zero trace values resulting from muting, those values will remain zero after the F-X deconvolution procedure.

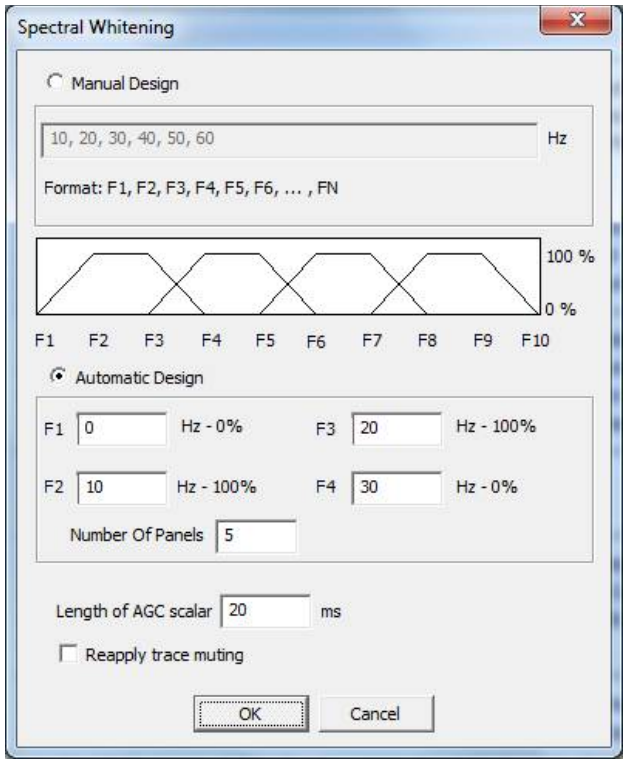
Spectral whitening

The spectral whitening procedure is used to expand and level the frequency spectrum of the seismic data.

Each input data trace is converted to the frequency domain, multiplied by the specified number of amplitude spectra, and then converted back to the time domain. The procedure produces a certain number of traces corresponding to the number of frequency bands specified in the module parameters, i.e. each trace is a filtered source trace in a particular frequency range. Automatic gain control (AGC) is applied to the resulting traces in the time domain. Then the traces with the AGC applied and the AGC factors are stacked to obtain the final trace and average AGC factor values. The true signal amplitude is restored by dividing the resulting trace amplitude by the average AGC factor value.

Module parameters:

- **Manual design** – manual specification of frequency bands. Four frequency values define one frequency band: F1 = 0%, F2 = 100%, F3 = 100% and F4 = 0%. Six frequency values define two frequency bands where the F3 and F4 frequencies are common for the first and second frequency band. For the second frequency band F3 = 0%, F4 = 100%, F5 = 100%, F6 = 0% etc. (see the screenshot).


Spectral Whitening
Manual Design
10, 20, 30, 40, 50, 60
Hz
Format: F1, F2, F3, F4, F5, F6, ... , FN
100 %
0 %
F1 F2 F3 F4 F5 F6 F7 F8 F9 F10
Automatic Design
F1 0 Hz - 0%
F3 20 Hz - 100%
F2 10 Hz - 100%
F4 30 Hz - 0%
Number Of Panels 5
Length of AGC scalar 20 ms
Reapply trace muting
OK
Cancel

- **Automatic design** – automatic division of the frequency band into the specified number of bands. The F1=0%, F2=100%, F3=0% and F4=100% frequencies define the "common" pass band. This band is divided into the number of bands specified in the Number of panels field.

- **AGC operator length** – value defining the length of the window (in ms) that will be used to calculate the gain factor.

- **Reapply trace muting** – restores zero values in trace amplitudes (resulting from muting) after running the filter.

Spectral shaping

This module is used to alter the shape of the input trace signal amplitude spectrum.

Module parameters and procedure description:

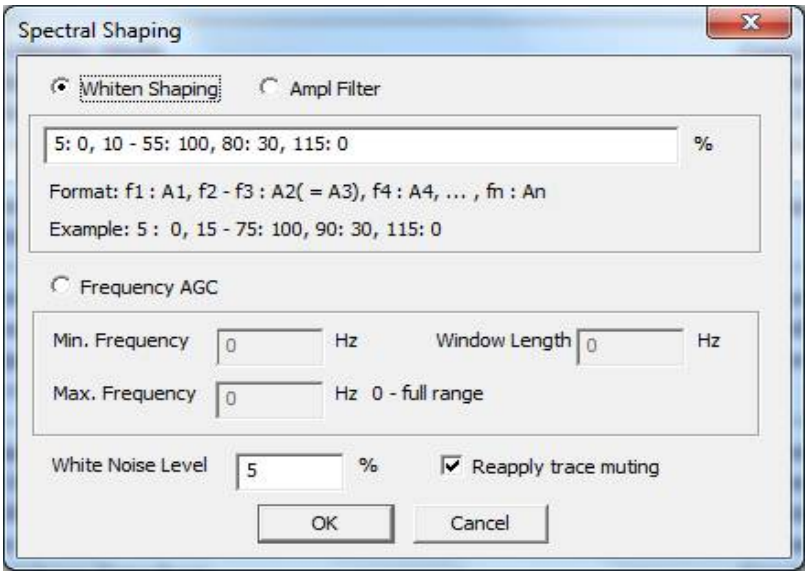
- **Whiten shaping** — defines the signal amplitude spectrum according to the specified shape, leaving the phase spectrum unchanged. The filter is created by specifying the Frequency:Amplitude value pairs. The amplitude values are specified in percent.

  The filter format is as follows:

  5:0, 10-55:100, 115:0

  The spectrum resulting from the application of the filter will be flat in the 10-55 Hz frequency range, will rise from 0 to 100% in the 5-10 Hz range, and will drop from 100% to 0 in the 100-115 Hz range.

- **Amplifier filter** – as a result of this procedure the current amplitude spectrum of the incoming traces will be multiplied by the contour specified in the shape parameters without changing the phase spectrum. The contour shape is defined in a similar manner – by specifying the Frequency:Amplitude value pairs.

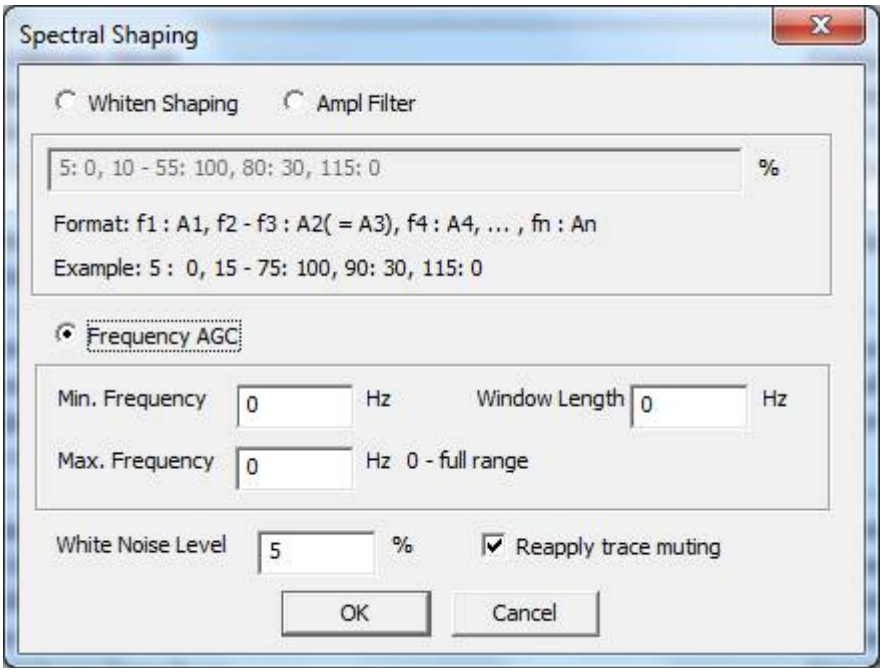


- **Frequency AGC** – this procedure performs automatic gain control in the frequency domain. The gain factor is calculated for each position of the window with the specified length in Hz sliding across the frequencies of the amplitude spectrum as an average of the absolute values throughout the window. The frequency step is obtained by dividing the Nyquist frequency by the number of trace samples. This factor is then applied to the window's central value. The amplitude spectrum variations are smoothed as a result of this procedure.

  Start frequency – start frequency to which the gain control will be applied.

  End frequency - end frequency to which the gain control will be applied.

- **Window length –** value defining the length of the window (in Hz) that will be used to calculate the gain.



- **White noise level** – defines the level of white noise to be added to the signal amplitude spectrum. Is specified in percent.

- **Reapply trace muting** – restores zero values in trace amplitudes (resulting from muting) after running the filter.

F-K Filter

This module is used to filter data in the F-K (frequency-wavenumber) domain. The user specifies a polygon in the F-K plane. This polygon defines a rejection or passing range for the filtering.

Before adding the module to the flow, you need to create a filter polygon. It is defined interactively in the Screen Display module window. You can view the two-dimensional F-K spectrum of a selected data fragment, construct a polygon for filtering in the F-K domain, preview the filtering results using the selected fragment as an example, and modify the parameters on the fly. After creating the polygon and choosing filtering parameters for the data fragment you can proceed to apply the filter to the entire data set in the flow using the F-K Filter module.

For the module to function correctly, the distances between the traces need to be approximately the same.

Brief F-K filtering theory

Two-dimensional (spatial) filtering algorithms are used to suppress noise waves or identify wanted waves with known properties. Spatial filtering is based on transferring the data to another domain (such as the frequency-wavenumber domain) by applying a mathematical transformation to the traces, filtering noise waves in that domain, and then converting the results back to the *(t, x)* domain.

The fk filter is an example of two-dimensional filters. It is based on twofold Fourier transform – a method of expansion the wave field into plane wave components. Each plane wave carries a monochromatic signal which propagates at a certain angle to the vertical. Events with the same inclination in the (t, x) plane are located on the same line in the radial direction in the (*f-k*) plane regardless of their position. As a result, for example, interfering signals inclined in the (*t, x*) plane may have no interference and may be successfully separated in the (*f, k*) plane by their inclination values. This allows eliminating the energy corresponding to noise waves (coherent line interference in the form of surface waves, channel waves, laterally dissipated energy obscuring true reflections) from the data. These types of noise are usually isolated from the reflected energy in the (*f, k*) space. (Yilmaz, 1989).

Viewing the F-K spectrum and setting up the filtering parameters

Viewing the FK spectrum

To view the FK spectrum, launch the Screen Display module.

Then click the **Tools/Spectrum/2D** spectrum menu item in the Screen Display window and select the data fragment for which the spectrum will be calculated with the mouse. To select the fragment, press and hold the left mouse button in a corner of the fragment, move the cursor to the opposite

corner of the fragment, and release the mouse button.

After you select the fragment, the FK spectrum window will appear. In this window you can view the FK spectrum for the selected data fragment, change the visualization parameters, construct a polygonal filter, save the constructed polygon to the program database, and test the filtering by previewing the results for the selected data fragment.

Setting up the TX and FK domain display parameters

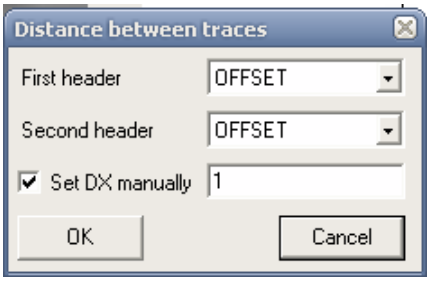
You can simultaneously view the data in the TX domain (left part of the window) and the FK domain (right part of the window). The TX domain has two axes: time in milliseconds (**Time**) and distance in meters (**Relative offset**). Distances are counted from the leftmost point of the window selected by the user using a fixed interval between the traces (the method of determining the interval based on the distance is described below).

The status bar at the bottom of the window displays information on the location of the cursor. If the cursor is within the TX domain, the status bar shows the current values of X (distance between the leftmost point of the selected fragment and the current location), T (time), and DX (interval on the X axis).

If the cursor is within the FK domain, the status bar shows the current values of K (wavenumber), F (frequency in Hz), dx/dt (coefficient of the inclination of line in the TX domain), and DX (horizontal interval).

Determining the interval on the X axis in the TX domain

For the FK spectrum to be displayed correctly, it is important to set the correct interval on the X axis in the TX domain. To set the interval on the X axis in the TX domain use Distance between traces option in the right part of the screen (or select the Parameters/Distance between traces menu item - the Distance between traces dialog box will open).



This dialog box allows you to specify the header fields for automatic calculation of the interval between the traces or set this interval manually.

To have the program automatically calculate the interval based on the specified header fields, select a pair of headers corresponding to the X and Y coordinates of the trace from the drop-down lists in the **First header** and **Second header** fields. These fields will be used to calculate the distance. The program calculates the interval as the arithmetic average of all distances between the traces in the selected data fragment.

If the user selects identical header fields, the program calculates the interval using only one header field. In this case the difference between all values of this header for adjacent traces will be averaged.

To set the interval manually, check the **Set DX manually** box and enter the value in meters.

Gain adjustments
To adjust the gain in the TX domain, move the TX Gain slider with the mouse.

Moving the slider to the right increases the gain, moving it to the left decreases the gain.

You can adjust the gain in the FK domain in a similar manner using the FK Gain slider.

Changing the scale in the TX or FK domain
To zoom into the fragment within the TX or FK domain, press the button and then select the fragment you need with the mouse. Image scroll bars will appear at the bottom and at the right side of the window.

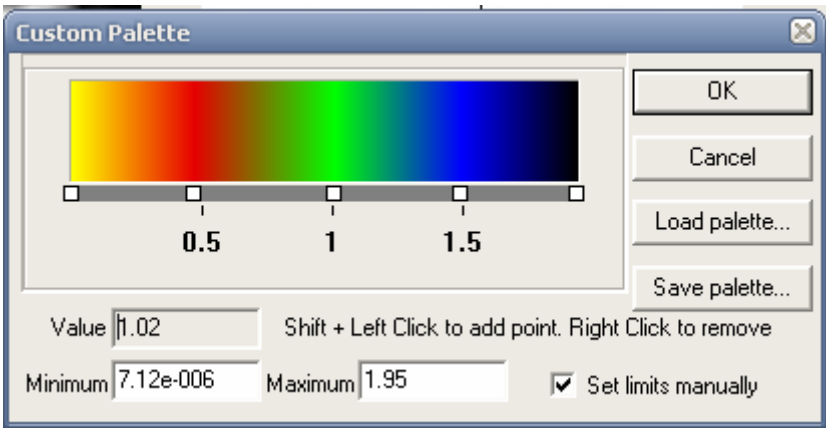
To revert to the original scale, press the button.

To gradually zoom out of the image (if it was previously enlarged), use the button.

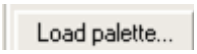
Setting up the image palette in the TX and FK domains
Use the following menu items to set up the image palette:

- **Parameters/ TX palette** – for the TX domain palette
- **Parameters/ FK palette** – for the FK domain palette

Since the palette setup dialog boxes are identical for both domains, only one is shown below.



The palette is defined as a set of points with the specified colors corresponding to certain values. Colors are linearly interpolated between the points within the maximum and minimum value range.

To load an existing palette, press the 

button.

This button opens a standard file selection dialog box that allows loading the palette from the disk.

The current palette can be modified manually. You can create, move and delete points, change the range of values used to build the palette, and view the values corresponding to a specific color.

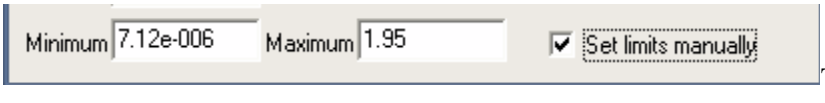
To change the position of a point, click and drag it with the left mouse button. The appearance of the palette will be automatically updated. When the mouse cursor hovers over a point or when a point is being moved, the current value corresponding to the selected point is shown in the **Value** information field.

To change the color of an existing point, double-click it with the left mouse button. A standard point color selection dialog box will appear.

To create a new point with the specified color, hold down the Shift key and left-click in the palette where you want the new point to be added. A standard point color selection dialog box will appear. A new point with the selected color will be added to the palette at this location.

Right-clicking a point deletes it from the palette.

By default the leftmost color of the palette corresponds to the minimum value present in the data, and the rightmost one – to the maximum value. The values corresponding to the palette boundaries can be modified manually. To do this, check the **Set limits manually** box.



The **Minimum** and **Maximum** fields will become active, allowing you to enter the new minimum and maximum values.

The edited palette can be saved to the database using the **Save palette** button.

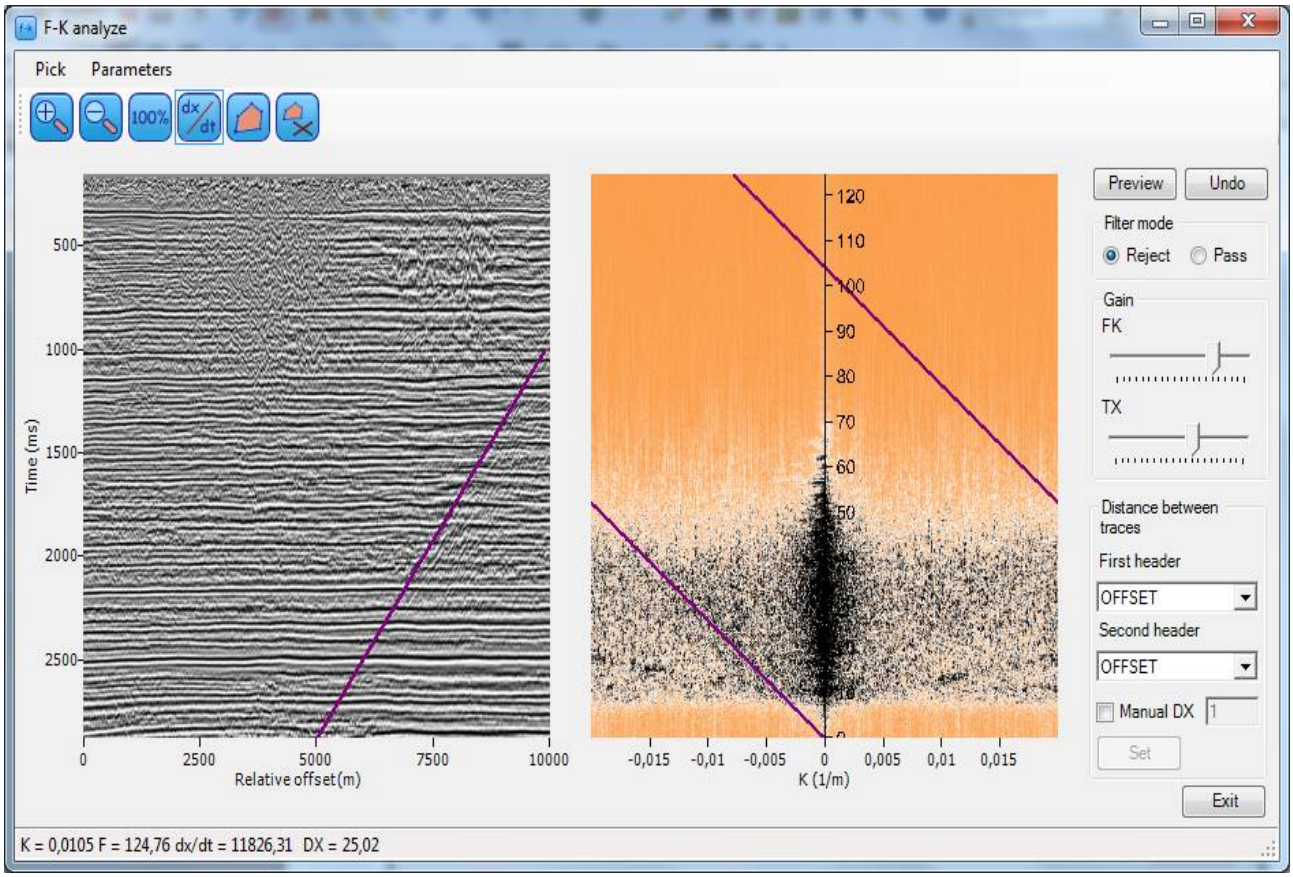
Determining apparent velocities (inclinations)

The line inclination defined in one of the domains will be automatically shown in the other domain taking the aliasing effect into account. This allows checking what an event in the TX domain corresponds to in the FK domain and vice versa.

To view the correlation between the inclinations in the TX and FK domains, press the button.

Place the cursor in the FK or TX field.

If you want to define an inclination in the TX domain and observe how it will be displayed in the FK domain, place the cursor in the TX domain. You will see a straight line originating from the upper left corner. The inclination of this line will be changed as you move the mouse cursor. To define a line originating from an arbitrary point, left-click in the TX domain where you want the origin of the line to be placed. The origin will be set, and the inclination will still be determined by the cursor movement. When working with inclinations, the inclination value dx/dt is displayed at the bottom of the screen. To fix the inclination line in the current position press once with the right mouse button. The possibility to fix the inclination line, which is corresponds to particular event, helps a lot in picking polygon procedure.



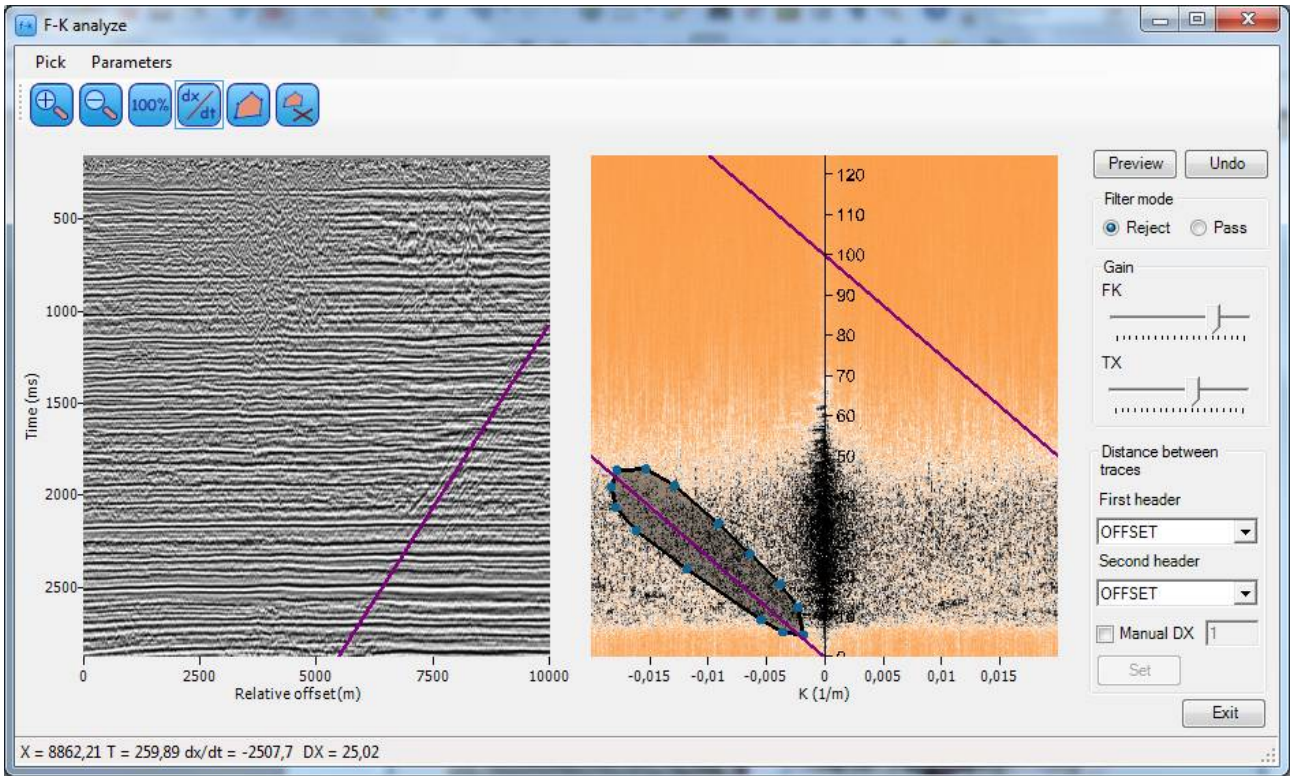
When you specify an inclination in the FK domain, the line origin will always be at the FK zero

coordinate. The line inclination is determined by mouse movement in the same manner as in the TX domain.

Creating and editing the filter

To apply FK filtering, you need to define a filter in the form of a polygon in the FK plane. The area inside the polygon can be rejected or accepted after filtering based on the selected filtering parameters.

- To create a polygon, select button (or select **Pick/New** menu item). After that you can interactively define the polygon vertices in the FK domain. To add a point (vertex), left-click in the FK field where you want the point to be placed.



- To move a point, place the mouse cursor over it, press and hold the right mouse button, and move the cursor to the desired location.
- To delete a point, double-click it with the right mouse button.
- To delete a polygon,press button.
- To return to the polygon editing mode, press button or select the **Pick/Edit** menu item.
- To save the filter to the project database, select the **Pick/Save polygon** menu item.
- To load a filter from the database, select the **Pick/Load polygon** menu item.
- Now that the filter is ready, you can preview the filtering results using the selected fragment as an example.

Previewing the filtering results

The polygon defined in the FK domain is used to filter the data. Depending on the filtering mode (**Filter mode**) the area inside the polygon will be either rejected (**Reject**) or accepted (**Pass**).

To preview the filtering results, press the Preview button.

To cancel the filtering and view the source data, press the Undo button.

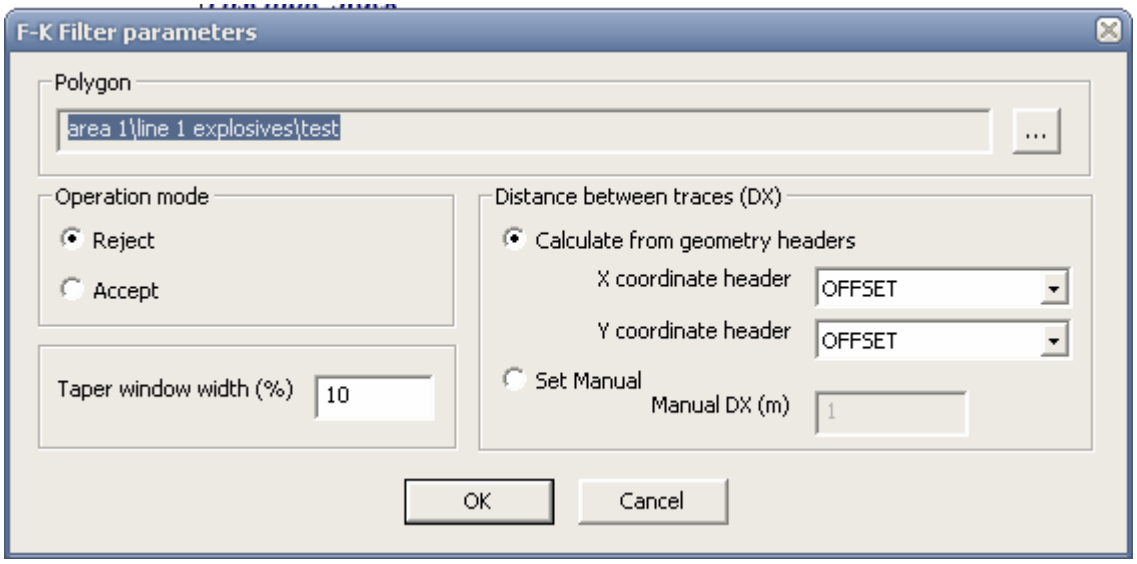
Mitigating edge effects during filtering

To avoid the "ringing" effect (caused by the Gibbs phenomenon) at the edges of the polygon during filtering, you can set the tapering window width as a percentage of the polygon area. Tapering will be applied to an area outside the polygon.

Applying the filter to the entire data set in the flow

Now that the filter has been interactively defined and saved to the database, it can be applied to the data in the processing flow. To do this, add the F-K Filter module to the flow.

The **F-K Filter parameters** dialog box will open.



Select the polygon that will define the filtering area from the project database. To do this, press the following button:

Select the filtering mode (reject or accept) and define the fixed interval between the traces either using the two header fields or manually. These dialog box parameters are similar to the ones in the interactive filter definition dialog box discussed above.

Trace Editing

Trace editing

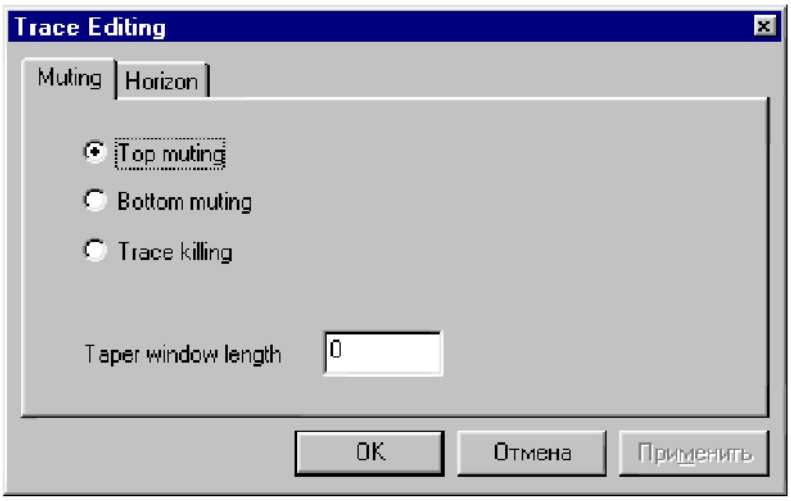
This module makes it possible to exclude invalid traces (dead) and/or record intervals (muting) from initial record.

Two factors make us carry out the editing:

- Initial data very often contains traces with extremely high amplitude values. It is caused by technical noise of different nature and has nothing in common with reflections from objects of medium. Traces identified by the user are being muted in order to eliminate their influence on processing results and interpretation quality. However, in order to preserve real horizontal scale they are not excluded from data.

- As a rule, the intensive noise which does not interfere with the desired trace fragment is registered at start and end time of seismic record. Almost in all types of processing they are subject to different transforms and significantly affect the result. Muting of some rectangular/rectangulars in a processed panel helps to neutralize this effect.

When this module is activated the following window containing two tabs: **Muting and Horizon** will open:



On the **Muting** tab, specify the type of editing:

- **Top muting** - allows muting of trace fragments from zero time to time defined by the user.

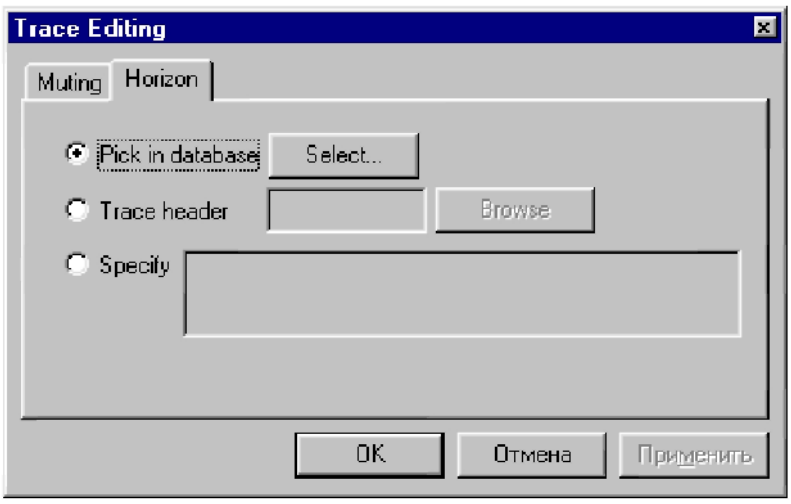
- **Bottom muting** - allows muting of trace fragments from time defined by the user up to maximum time.

- **Trace killing** - allows muting of separate traces defined by the user.

When selecting the **Top/ Bottom muting** the **Taper window length** window becomes active.

Trace tapering multiplies traces by symmetrical function of the window with cosine edges. The length of the window equals the length of the trace. In the Taper window length, specify the window length percentage where the function will be expressed as a straight line, i.e. will not change traces amplitudes.

When the **Horizon** tab is activated the following window opens:



where it is required to specify the pick for muting. There are three ways to do this:

- **Pick in database** - specify the pick from database. To do this, activate this option, click the **Select...** button and in the opened dialog box select the desired pick saved in the project database.

- **Trace header** - specify the pick from trace header field. To do this, activate this option, click the **Browse...** button and in the opened dialog box select the desired header.

- **Specify** - specify the pick manually in the **Specify** field.

An example:

CDP

0-50:500,70:300

In this example:

CDP - header field name;

0-50 - field values, in this case these are CDP points numbers;

500 - time, etc.

Between specified positions the muting times will be interpolated horizontally.

Trace length (Trace length changing)

This module is applied to change the number of samples in the traces of the flow. In case when the new value samples number is smaller than the old one then traces are being cut off. Otherwise, they are supplemented with zeroes to comply with a new length. This procedure can be useful in the following cases:

- When you need to reduce the data from different input sequences to one number of traces
- When you need to add zero samples to traces in order to provide space for static shift
- When you need to cut unneeded trace fragment off in order to increase processing speed

When this module is activated the following window appears:

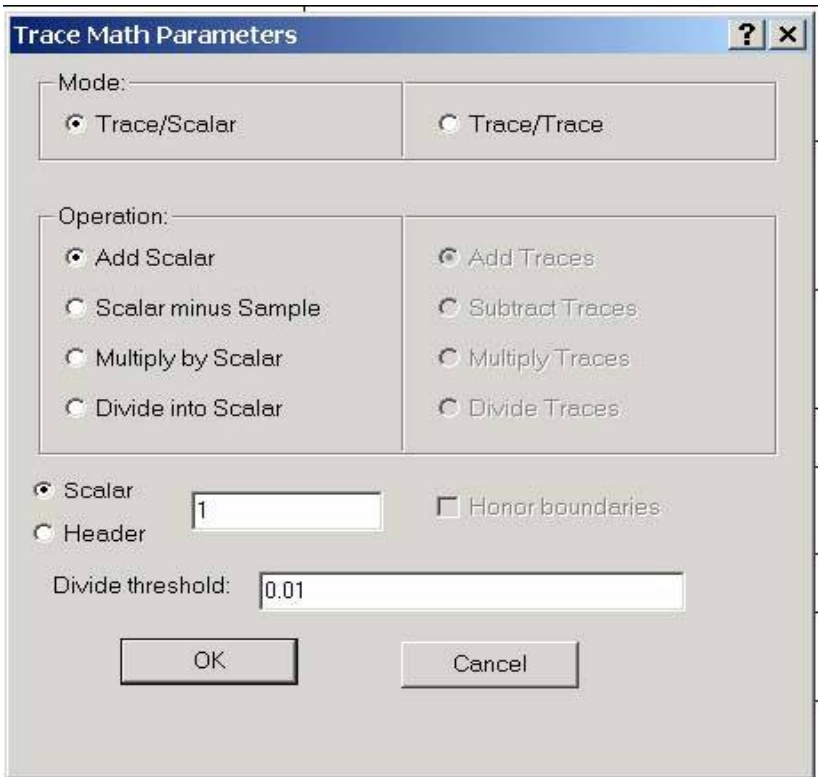
where, in the **New trace length** field, specify a new length of trace (expressed in ms).

Trace Math

The Trace Math module is applied to conduct arithmetic operations with adjacent traces or with traces and scalar.

There are two ways to carry out the module: trace with trace and trace with scalar. When the first method is used, every trace at the module output is a combination of two traces at the input. If the number of traces is an odd number then the last trace will remain unchanged.

When starting the module the following dialog box will open:



where, **Mode** - selection of module execution method: **Trace/Scalar** - trace with scalar or **Trace/Trace** - trace with trace.

Trace/Scalar method

In the **Trace/Scalar** method in the **Scalar** field, specify the scalar value which will be applied to accomplish selected operation. You can also use for every trace a value written in one of its header fields. To do this, select the **Header** option and, in the same field, replace the value by the header

field name.

The following operations are possible between the trace and the scalar:

- **Add Scalar** - add the scalar to every sample;
- **Scalar minus Sample** - subtract every sample from the scalar;
- **Multiply by Scalar** - multiply every sample by the scalar;
- **Divide into Scalar** - divide the scalar by every sample;
- **Reverse Trace (Trace/Scalar) –** turns each trace back to front in time (i.e. the first sample becomes the last one);
- **Cross Correlation (Trace/Trace) –** calculates cross correlation within the pairs of neighboring traces (as a result the number of traces is halved);

The last option requires specification of threshold value in the **Divide Threshold** field. In case when the absolute value of the sample is smaller than the threshold defined by the specified parameter then division will not be accomplished.

Trace/Trace method

The following operations are possible:

- **Add Traces** - sum traces by samples;
- **Subtract traces** - subtract traces by samples;
- **Multiply traces** - multiply traces by samples;
- **Divide traces** - divide traces by samples;

The last option requires specification of threshold value in the **Divide Threshold** field. In case when absolute count values are smaller than threshold one defined by specified parameter then division will not be accomplished.

Deconvolution

Deconvolution (Deterministic deconvolution)

This module is meant to accomplish deterministic deconvolution. As a matter of fact, this module can accomplish not only the deconvolution but any other spectral operations with traces and specified signal as well.

Theory

Deconvolution is an inverse filtering of a signal with defined source pulse. The registered reflected signal (trace) is interpreted within linear model defined by convolution integral. Within the framework of this model the frequency characteristic of a trace is a product of characteristic of receiving-recording channel and frequency characteristic of the medium. The latter includes distortions induced by the medium which are actually the interpretation errors of observed data (for example, multiple reflections, ghost-waves formed on the Earth's surface and the waves similar to them). The essence of the deterministic deconvolution consists in that the frequency characteristic of reflected signal (trace) is divided by frequency characteristic of initial impulse (i.e. the impulse of receiving-recording channel). Ideally, when doing this, obtain the frequency characteristic of the medium which, when converting it to time domain (if you do not take into consideration stated above distortions caused by the medium), will return reflection coefficients section at the output. In practice due to some reasons it is impossible. However the deterministic deconvolution allows significant narrowing of the signal by increasing data resolution ratio.

Before deterministic deconvolution application, determine the impulse of source for inverse filtering (signature). This impulse should be saved in file. This file must contain the sequence of real numbers in R4 format (IEEE). To obtain signature you can use the procedure described below.

- Select a fragment of profile with traces you would like to use for signature determination.

- Pick the reflected signal which will be used for deconvolution via the picking function of the *Screen* ***Display*** module. To speed up the operation you can use the automatic pick mode. The pick must be saved either in the database (the **Tools|Pick|Save** module command) or in the text file (the **Tools|Pick|Export...**module command).

- Reduce the impulses to common time via the **Apply Statics** module with static corrections obtained at a previous stage.

- Sum corrected traces via the **Ensemble Stack** module.

- Save the obtained trace into file via the **Data Output** module having the following parameters:

Format: User-defined

User-defined format parameters:

File passport: 0

Trace Passport: 0

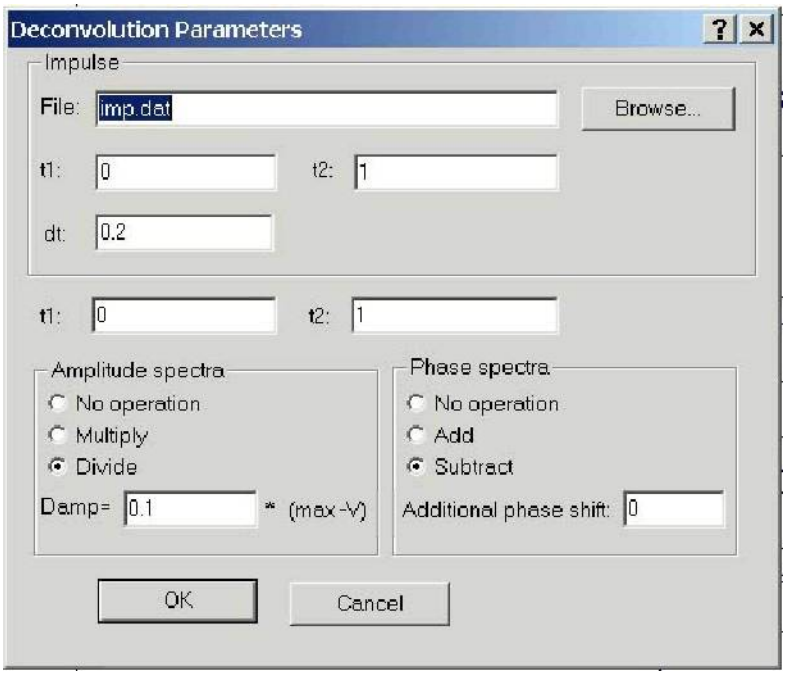
Trace points: the value is selected in accordance with impulse length (including impulse delay). You can use the number of samples of input data flow.

Data format: R4

From x, to x, From t, to t to be set in accordance with the values containing impulse.

Parameters

When this module is activated the following window appears:



Impulse group of parameters defines the signature:

- **File** - a file that contains the trace with signature. To define the file name, click the **Browse...** button or enter the name and the path manually;

- t1, t2 - start and end time of signature in trace (it is considered that the first sample has zero time);

- **dt** - sampling step of the trace with signature;

Deconvolution parameters:

- **t1, t2** — start and end time of vertical window to which the deconvolution will be applied (usually it is set so to include the hole trace time range).

The **Amplitude spectra** group of parameters defines what operation to be applied to amplitude spectrum of every trace and amplitude spectrum of impulse:

- **No operation** - operation with amplitude spectrum is not carried out;

- **Multiply** - amplitude spectrum of every trace will be multiplied by amplitude spectrum of signature;

- **Divide** - amplitude spectrum of every trace will be divided by amplitude spectrum of the signature. The operation of spectrum division is unstable. That is why the damping factor **Damp** (often called "white noise level") should be specified. This value is added to the spectrum of impulse before division is accomplished.

The **Phase spectra** group of parameters defines the type of operation applied to phase spectrum of every trace and to impulse phase spectrum:

- **No operation** - operation with phase spectrum is not carried out;

- **Add** - signature phase spectrum will be added to phase spectrum of every trace;

- **Subtract** - signature phase spectrum will be subtracted from phase spectrum of every.

- **Additional phase shift** - This is a time shift of the whole trace which accomplished after all selected operations.

Different combinations of operations applied to amplitude and phase spectra of data are the following:

Amplitude spectrum transforms	**Phase spectrum transforms**	**Meaning**
No transforms	Subtract	Phase deconvolution
Multiply	Subtract	Correlation
Divide	Subtract	Deconvolution
Divide	No transforms	Zero-phase deconvolution
Multiply	No transforms	Optimum filtering
Multiply	Add	Convolution

Custom Impulse Trace Transforms

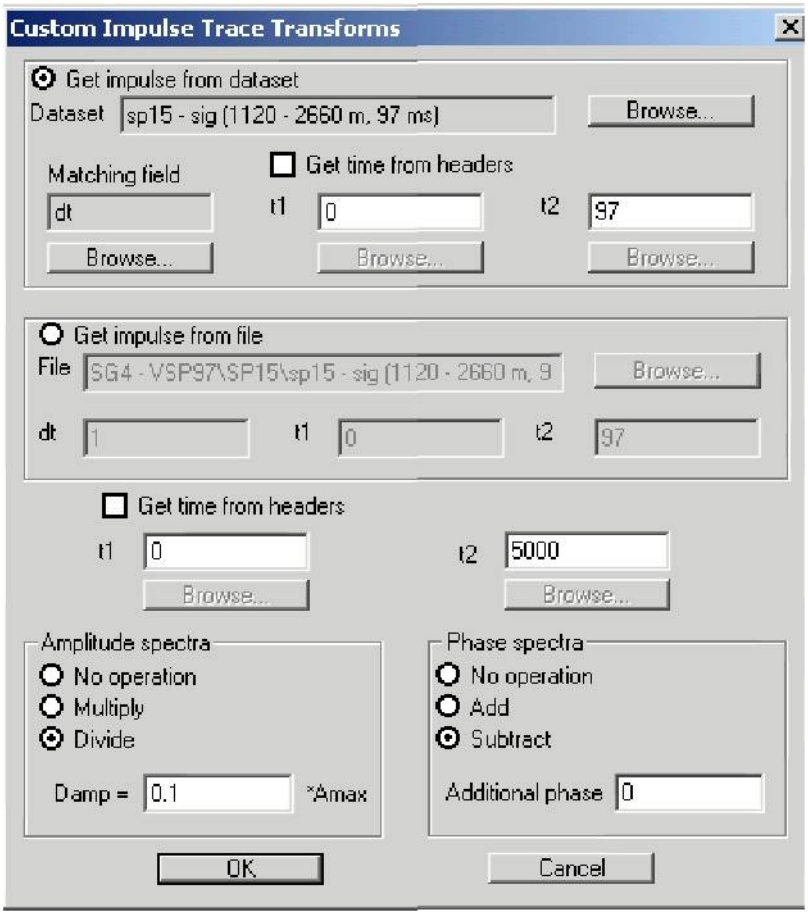
Theory

This module conducts a set of operations of phase and amplitude spectrum of trace and impulse (set of impulses).

Different combinations of operations of phase and amplitude spectrum are the following:

Amplitude spectrum transforms	**Phase spectrum transforms**	**Meaning**
No transforms	Subtraction	Phase deconvolution
Division	Subtraction	Deconvolution
Division	No transforms	Zero-phase deconvolution
Multiplication	No transforms	Optimum filtering
Multiplication	Addition	Convolution
Multiplication	Subtraction	Correlation

Parameters



The **Get impulse from dataset** group of parameters allows selection of the impulse (impulses) from the project database:

- **Dataset** - a dataset from which the impulses can be selected. To select a dataset, click the **Browse...** button.

- **Matching field** - a field according to which the trace selection is carried out, i.e. according to the value of this field the trace searching within selected dataset is accomplished for every trace.

- t1, t2 - the start and end time of time window for the impulse. It is possible to use the values of header fields of seismic traces as t1 and t2 but in this case field selection is accomplished by clicking the **Browse...** button (to do this the **Get time from headers** mode should be chosen).

The **Get impulse from file** group of parameters:

- **File** - is a file containing trace with impulse. This file must contain the sequence of real numbers in R4 format (IEEE). To select a file, click the **Browse...** button.

- t1, t2 - the start and end time of time window for the impulse. (The first trace sample with impulse is considered to be the time zero).

- **dt** - sampling step in trace with impulse.

General parameters of operation:

- t1, t2 - the start and end time of time data window in which the selected spectra transform will be carried out.

- It is as well possible to use header field values of seismic traces as t1 and t2. In this case selection of field can be done by clicking the **Browse...** button (to do this the **Get time from headers** mode should be chosen).

The **Amplitude spectra** group of parameters defines what operations to be done with amplitude spectrum:

- No **operation** - operation with amplitude spectrum is not carried out;

- **Multiply** - amplitude spectrum of every trace will be multiplied by impulse amplitude spectrum;

- **Divide** - amplitude spectrum of every trace will be divided by impulse amplitude spectrum. The

operation of spectrum division is unstable. That is why the damping factor **Damp** (often called "white noise level") should be specified. This value is added to the spectrum impulse before division is accomplished.

The **Phase spectra** group of parameters defines what operations to be done with phase spectrum:

- No **operation** - operation with phase spectrum is not carried out;
- **Add** - impulse phase spectrum will be added to phase spectrum of every trace;
- **Subtract** - impulse phase spectrum will be subtracted from phase spectrum of every trace.
- **Additional phase** - This is a time shift of the whole trace which accomplished after all selected operations.

Predictive Deconvolution

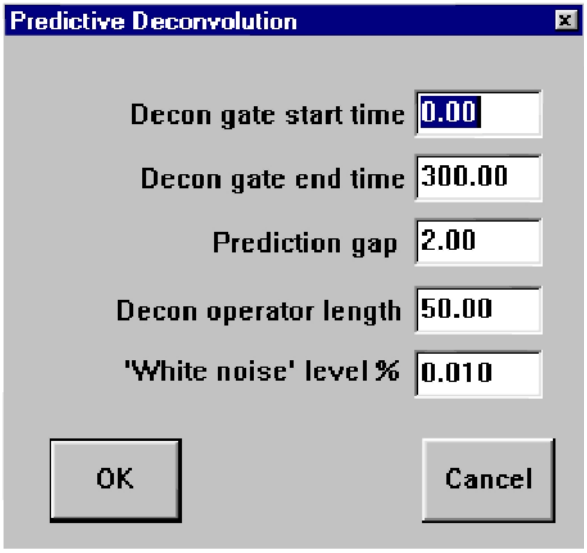
This module is applied in order to carry out predictive deconvolution.

Theory

Predictive deconvolution is applied to remove coherent noise and to increase resolution ratio. The reflected signal (trace) is interpreted within linear model assigned by convolution integral. Within the framework of this model the frequency characteristic of trace - is a product of characteristic of receiving-recording channel and frequency characteristic of the medium. The later includes distortions caused by the medium which are actually the interpretation errors of observed data (for example, multiple reflections, ghost-waves formed on the Earth's surface and the waves similar to them). Within the process of predictive deconvolution the operator of linear filter applied for above mentioned distortions removing is calculated. To do this, in order to solve Viner-Hopf equation, Viner-Levinson algorithm with white noise adding is used and in order to reduce prediction error to minimum the least-squares method is used.

Parameters

When this module is activated the following window appears:



- **Decon gate start time -** start time for deconvolution operator

- **Decon gate end time** - end time for deconvolution operator construction. It must be greater than **Decon gate start time,**

- **Prediction gap** - Prediction operator gap to be applied to input data expressed in ms,

- **Decon operator length** - deconvolution operator length expressed in ms. It defines the length for autocorrelation function calculation,

- **"White noise" level** - "white noise" level expressed in percents. This percentage of preliminary whitening defines what percentage of white noise must be added to the original impulse characteristic.

NOTE: Also, in order to increase record resolution the **Predictive Deconvolution** module can be used for spiking deconvolution. To do this, in the **Prediction gap** field, specify prediction operator gap as equal sampling step and in the **Decon operator length** field, specify deconvolution operator length as equal at least the length of outgoing impulse.

Surface-Consistent Deconvolution

Parameters

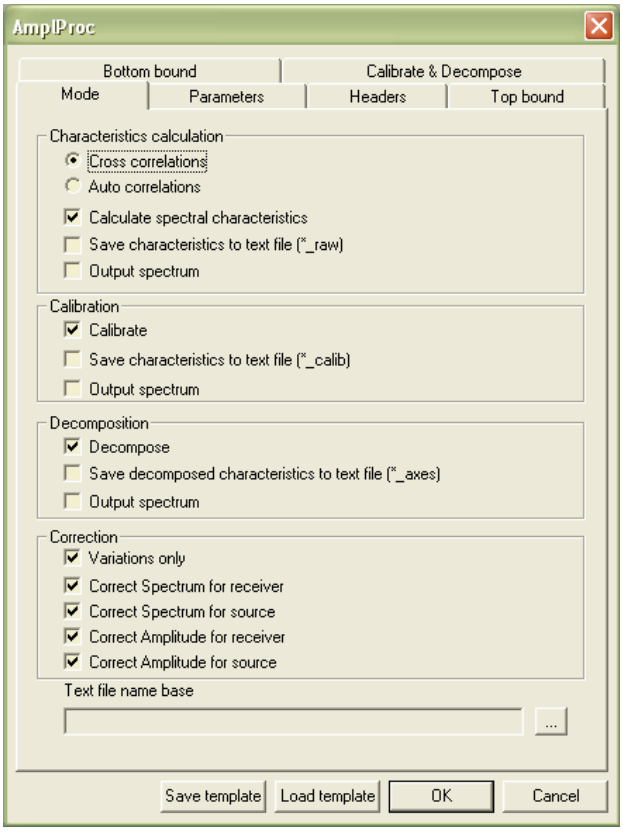
- **Mode Tab**

Calculation. Amplitude characteristics (both spectral and scalar) can be evaluated either by means of cross-correlation function or auto-correlation function. The first method has preferences because of better resistance to random disturbances. By default program calculates only scalar amplitude, spectral amplitudes and their output to standing alone dataset or txt-file are optional.

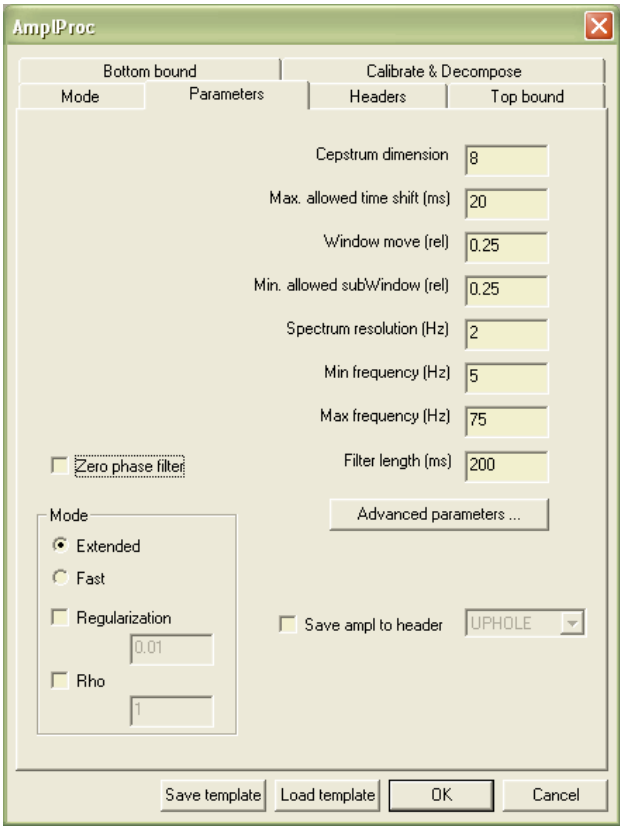
Calibration means 2-factor model fitting. The corresponding tolerance parameter should be set in "Calibrate & Decompose" section. Optionally you can view the resulting calibrated spectra or save it to file in text format.

Decomposition means recovering both factors for 2-factor model in order to use them for separate filtering. There exist also possibility to consider these factors as signals and save them either to standing alone dataset or to text file.

Correction means construction of the filter which damps the corresponding factor effect and application this filter to the trace record. Such a filter can be implemented either by means of true amplitudes or by means of their variations versus global "average" value (this average value is defined as an exponent applied to the mean value of logarithms of non-calibrated amplitudes for all dataset).



- **Parameter Tab**



Cepstrum dimension and Mode. Amplitude spectrum and its logarithm can in given frequency segment can be approximated by finite linear combination of functions belong to some base family. The number of functions under consideration is "Cepstrum dimension" parameter, approximation method is defined by "Fast" or "Extended" buttons. When "Extended" method is on there exist an additional tuning parameter Rho, in general terms the larger the value of Rho the more smooth is spectral amplitude function. "Regularization" parameter controls relative value to be add to amplitude spectrum.

Max.allowed time shift (ms) is an estimation of hodograph time shift for the two neighbouring traces (when considered in initial sorting).

Window move (relative measure). In order to evaluate covariance function the moving window size and the corresponding shifts should be defined. The entry input define ratio of the size of elementary shift of the moving window to the window length.

Minimal allowed subWindow (relative measure). When calculating the covariance function over its domain (this domain is assumed to be assume symmetrical segment $[-\tau, \tau]$) the different displacements of the moving window are necessary. For any displacement the subwindow (= intersection of the two windows) is defined, the entry value is the lower bound of such a

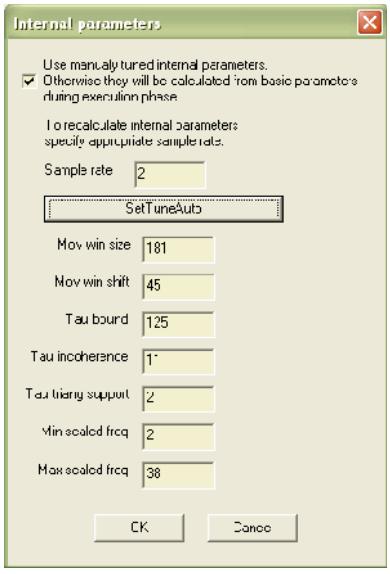
subwindow in relative units.

- **Spectrum resolution.** Covariance function domain can be defined in terms of desired spectrum resolution.

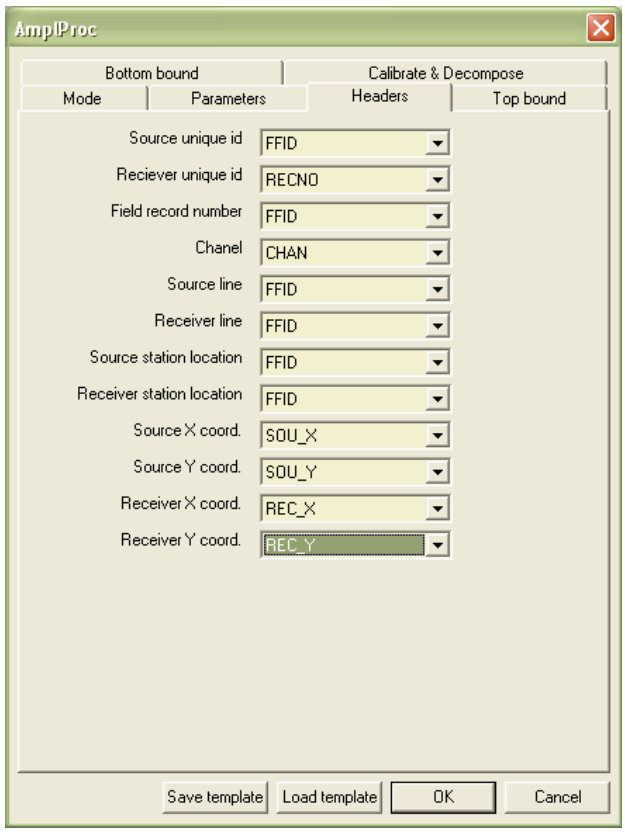
- **Minimal and maximal frequencies.** These are the left and right bounds of amplitude spectrum under consideration.

- **Filter length and zero phase checkbox.** The must be an agreement between source signal length and filter length. By default filter is Minimal Phase Filter, special case is Zero Phase Filter.

- **Advanced parameters.** Listed above entries together with sampling rate of trace record are sufficient to define in absolute units the moving window size and its dynamics. Nevertheless the user may disagree and may want to manually set internal absolute parameters (**not recommended!**) . To make this settings, which are not at all safe from calculus internal limitations, you should enter first the sampling rate to child window entry, then click "SetTuneAuto" to see the corresponding absolute values:

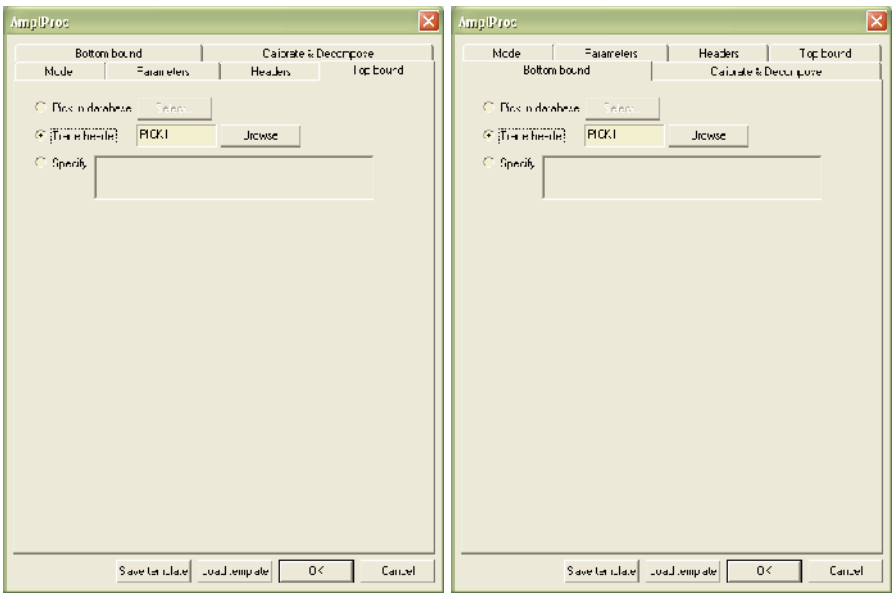


- **Headers Tab.** The 2-factor model needs indexation, the corresponding "Source unique id" and "Receiver unique id" are two indices under consideration. Other entries are reserved for the extended version of the module.



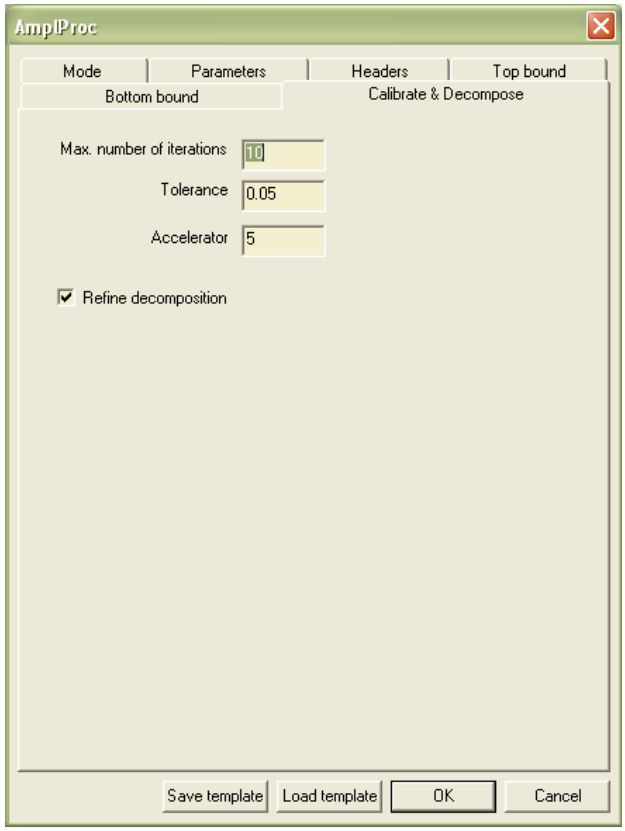
- **Top Bound and Bottom Bound Tabs**.

These are standard dialog windows for selection special time bounds within traces.



- **Calibrate & Decompose Tab.** Here Tolerance constant for the approximation can be set. In order to make calibration execution faster you can reduce the number of conditions in consideration (generally the number of such conditions is of the order of square the number of traces) by division to the "Accelerator" value. Since calibration is an iterative process we also reduce the number of iterations, calibration will be done when either "Maximal number of

iterations" or "Tolerance" parameter exceeded. Decomposition quality also can be improved by means of similar iterations, however there exists a kind of "uncertainty relation" for given difference ε (between 2-factor model and calibrated data) and the number N(ε) of entries where this difference is exceeded. "Refine decomposition" checkbox is used to decrease maximal observable difference max(ε).



Nonstationary predictive deconvolution

If subtrahend field is raw field, shifted by one sample, and it is input to an algorithm of adaptive subtraction of wave fields (realized in module **Wavefield Subtraction**), then the calculated filter will be considered to be a predictive-error filter. Thus the procedure of subtraction gets the significance of predictive deconvolution. If we set non-zero number of special basis functions, this deconvolution equivalent can theoretically correct the wavelet shape versus time variation, caused by different reasons, and that's why the amplitude spectrum equalization can be better, then as a result of conventional deconvolution. Basically you can input shifted data into wavefield subtraction algorithm independently, but in this case the band pass transform will be performed after a shift, that is not quite correct. For that reason and as well for convenience, a separate module

is realized in the program, in which the process works as non-stationary deconvolution, described above. In given mode only one wavefield is input, the field to which we have to apply predictive deconvolution. The subtraction algorithm is identical to the wave fields' subtraction algorithm, realized in the module Wave field subtraction.

That's why the description of parameters, that are in general similar to parameters of the indicated module, will be done in the context of subtraction of one wavefield from another one.

Remarks on using the subtraction algorithm in the mode of non-stationary deconvolution

- Non-stationary deconvolution is attractive because it can use statistics on a large window; that's why there is no need to divide data field into windows. However in some cases there is no possibility to avoid this.

- In the given mode the functions of multiplication are responsible for correction of non-stationary nature of seismic response.

- In the given mode revoking components have no sense, as the filter should be purely predictive.

- Processing is very sensitive to regularization parameter in this mode.

- As there is a shift on one sample, using it when subtracting neighboring traces can cause a big error. That's why the option of spatial channel number in this module is not realized.

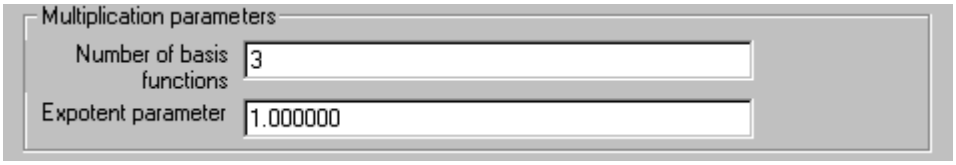
- The working frequency bandwidth should be selected, to avoid zeros in spectrum, if possible.

Parameters

- **Input data.**

Zero phase input data – flag that indicates that the input data are zero-phase. The given option works only is stationary mode. At the same time the filter is recalculated into its' zero-phase analogue.

- **Multiplication parameters.**



The given parameters' set describes the characteristics of the basis functions that are used in multiplication of subtrahend field traces.

The process algorithm contains, in particular, the calculation of the shaping filter using the least squares technique, so that the filtered subtrahend field in the sense of the least squared deviation would be close to the initial field. This filter is considered to be a multichannel filter, while the additional channels for its calculation and subsequent convolution are taken from the raw trace, by means of adding parametrized non-stationarity in time domain. More particularly, the assignment of each multiplication function adds a channel, formed by the multiplication of the raw trace by function:

$t^{\alpha \cdot n}$, where

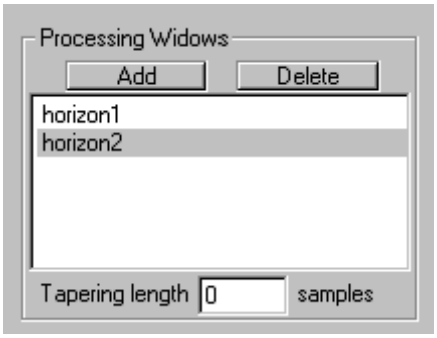
n – number of multiplication function, while alpha – is so called **Exponent parameter**, used for fine tuning, and usually is equal to 1.

Thus, without consideration of **Exponent parameter** we can say that if **Number of basis function** is 0, the shaping filter is considered to be single channel; if this parameter is equal to 1, the trace, multiplied by time is added to the shaping; if it is equal to 2, a trace with squared non-stationarity is added etc.

The more we assign multiplication functions the better we subtract one field from the other; but we have to take care, as in this case we can subtract undue value from the raw field. You should come to a compromise, and the number of multiplication functions is selected individually proceeding from the processor's experience in each case. You should also keep in mind that the extent of subtraction depends on the other parameters, particularly the filter length and the length of working

window. The more the filter length and the less the window length, the better we subtract (we imply that we subtract undue value).

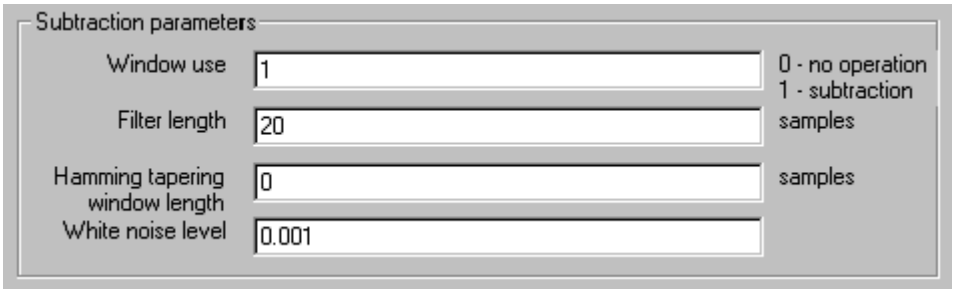
- **Processing windows**



Considering the records' nature, sometimes the processing should be done in different windows individually. The procedure supports the division by windows using picks' assignment – processing windows' boundaries. Thus if there is no boundary in the list, we conclude that there is only one window from the start to the end of trace. If only one pick is assigned, the whole processing area is divided into two windows: one – from the start to the pick, another from the pick to the end of trace. If you assign two picks, we have three processing windows respectively, etc. The boundaries of picks are added/removed using the buttons **Add** and **Delete**, the current set of boundaries is represented in the list. The user should insure that the boundaries do not intersect, since the program behavior is unpredictable over intersecting boundaries. As well you shouldn't use too narrow windows, as there will be subtraction to zero that is no good. However sometimes it is useful to set a narrow window and mark is as non-active (i.e. there will be no subtraction in this window); in such a way you can perform a so-called muting of areas that are unsuitable for subtraction.

Tapering length parameter sets an area of results' stitching in different windows. It is measured in samples

- Subtraction parameters.



The given set of parameters characterizes the parameters of shaping filter.

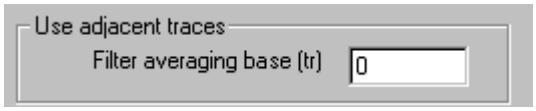
Window use – sets the sign of whether to perform subtraction in the given window or to keep unchanged the raw field. (1 – subtraction in the window should be performed, 0 – should not be performed, correspondingly).

Filter length - sets the length of shaping filter, in samples.

Hamming tapering window length – specifies the size of Hamming window in samples. If there is 0, the Hamming window is out of use.

White noise level – regularization parameter. Fractional additive to the main diagonal of autocorrelation matrix while solving the system, sometimes for some reason is called white noise level.

- Use adjacent traces.

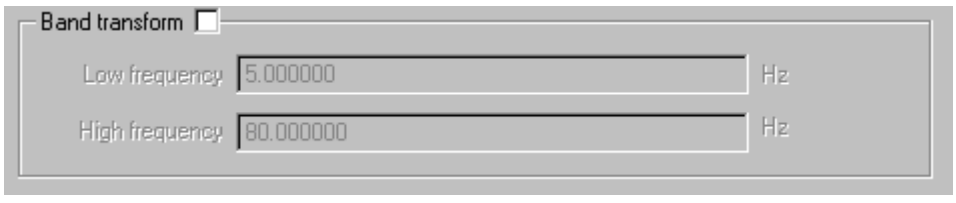


Filter averaging base – sets the arm of averaging base of autocorrelation and cross-correlation matrixes during filter calculation. In other words the shaping filter is averaged by a certain number of traces.

- Other parameters.

Accuracy – accuracy of calculation. Parameter is used, in particular, while comparing the traces' energy before and after filtering.

- Band transform.



You can perform filtering, within full bandwidth as well as within a limited bandwidth. The button

Band transform is designed to toggle between the modes.

When you use the mode of limited bandwidth, re-sampling is carried out, more precisely, band transform (see report attachment) of the data from the specified bandwidth to the whole area. An inverse band transform is applied to the result after carrying out the subtraction. This stratagem is used because the algorithm is designed for the work in full frequency range, while practically for some reason it is preferable to use a limited bandwidth. **Low frequency** and **High frequency** parameters assign low and high frequencies of the used bandwidth, correspondingly

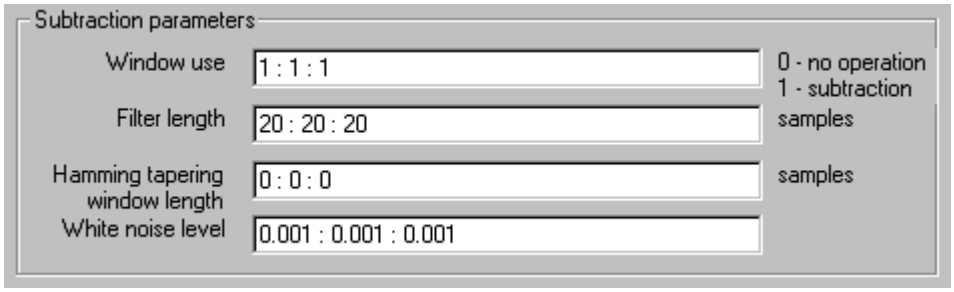
Remarks on parameters assignment in several processing windows

Due to properties with depth variation it is helpful to divide the whole data area into processing windows, and to use specific processing parameters for each separate window. It is admitted by a number of parameters of the module, namely:

Number of basis function, Exponent parameter, Windows use, Filter length, Hamming tapering window length, White noise level, Low frequency and **High frequency.**

In this case the values are specified using separation character ':'.

For example, subtraction parameters for three processing windows can be written as follows:



Static Corrections

Calculate Statics

The module is designed for the calculation of datum static corrections for PP and PS waves. The values of corrections can be calculated for each trace for PP, as well as for PS waves. The acquired values are written down to the header fields indicated by the user.

Attention: The following header fields should be present in order to work in the project: SOU_STAT, REC_STAT, SOU_STAT1, REC_STAT1.

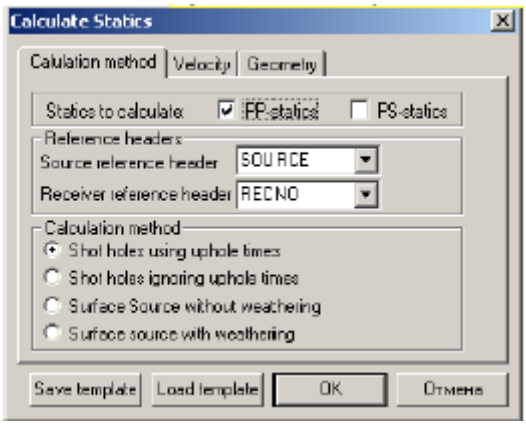
The traces from the flow with any sort order are input in this module. The source and receiver positions are determined from the header fields indicated by the user. The altitudes and the source depths, the altitudes and /or uphole time values are taken from the text files or from the header fields indicated by the user. Vp and Vs velocities (in weathering layer and the layer from the weathering zone to the final datum) are taken from the editing field in the parameters window and can vary in lateral direction.

(uphole time – travetime from the source in the borehole, upwards to the earth surface. The borehole source is supposed to be places below the weathering zone (low-velocity layer or weathering).)

The unchanged traces with the initial sort order are the output from the module. Shot and receiver static corrections for PP waves are written to the header fields SOU_STAT and REC_STAT subject to user selection and to the header fields SOU_STAT1 and REC_STAT1 - for PS waves.

Module parameters

The following dialog window including three tabs appears when you activate the module:



Select the calculation method of static corrections on the tab **Calculation method**:

- **Statics to calculate** – indicate the type of waves to calculate corrections

PP-statics - option that indicates that corrections for P-waves will be calculated.

PS-statics - option that indicates that corrections for PS-waves will be calculated.

- **Reference headers** – you should indicate the header fields, including the stake numbers with source and receiver positions.

Source reference field – header field that is used to determine the source position (source stake number), SOURCE by default.

Receiver reference field - header field that is used to determine the receiver position (receiver stake number), RECNO by default. (We believe that the stake numeration is the same for sources and receivers, i.e. if SOURCE=10, then the source was situated at the stake number 10, while RECNO=10, means that the receiver was at the same place)

- **Calculation method** – method of corrections calculation.

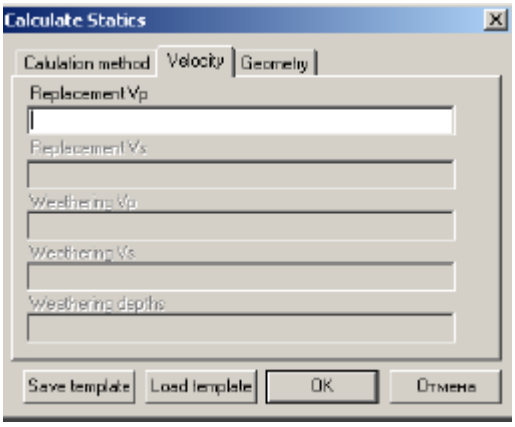
Shot holes using uphole times – uphole times are used for corrections calculation (by default).

Shot holes ignoring uphole times – the values of uphole times are ignored. The traveltime through the weathering zone is determined from the values Weathering Vp and Weathering Vs.

Surface source – the sources are situated on the surface, uphole time is not used, the effective velocities Replacement Vp and Replacement Vs are used for calculation.

Surface source with weathering – the sources are situated on the surface, uphole value is not used, the depth of the weathering layer and velocity are set for profile stakes (similar to the borehole depth).

Further you have to set velocity values (as well as thickness of weathering zone, if it is used) that will be used for statics calculation on the **Velocity** tab:



A set of fields that are available in the tab, depend on the selected options in **Calculation method**. You have to set the values to all accessible fields.

The values can change laterally (they are interpolated linearly between the stakes). The syntax of the values on the tab is the following:

number_stake : value, number_stake – number _stake : value

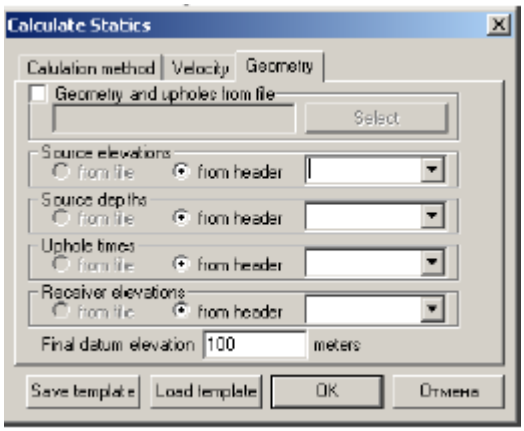
- **Replacement Vp** – the values of Vp associated with the stakes numbers are indicated in the editing field.

- **Replacement Vs** - (accessible if the option PS-statics is on) the values of Vs associated with the stakes numbers are indicated in the editing field.

- **Weathering Vp** - (is accessible, if you choose **Shot holes ignoring uphole times** or **Surface source with weathering** in the field **Calculation method**) the values of Vp in the weathering layer are assigned in association with the stake numbers in the editing field.

- **Weathering Vs** - (is accessible, if you choose **Shot Holes Ignoring Uphole Times** or **Surface source with weathering** and the option **PS-statics** is toggled on in the field **Calculation method**) the values of Vs in the weathering layer are assigned in association with the stake numbers in the editing field.

- **Weathering depths** - (is accessible, if you choose **Shot holes ignoring uphole times** in the field **Calculation method**) the values of Vp in the weathering layer are assigned in association with the stake numbers in the editing field.



And finally, you should indicate the source of each geometry parameter on the tab **Geometry** that you are going to use when calculating statics, as well as the level.

- **Geometry and upholes from file** – option that allows indicating the text file (~A-format, see Borehole Loading and visualization.../ File formats), from which the altitude/depths and upholes values are read. If the option is toggled on, the **Browse** button for selecting files becomes available. There is a possibility of reading several (or all) important geometry parameters from the corresponding file columns (the column names for each parameter are enclosed below). If the option is toggled off, all geometry values should be read from the trace headers.

- **Source elevations**

  - **from file** – the option is available, if **Geometry and upholes from file** is toggled on. The values will be read from the SE column of the selected file if you choose the option.

  - **from header** – when you select this option you have to choose the header field, containing the corresponding values from the pop-up window on the right.

- **Source depths** – (available in all cases, except the **Calculation method: Surface Source**)

from file – the option is available if the **Geometry and upholes from file** is on. When you select this option the values will be read from the SD column of the chosen file.

from header - when you select this option you have to choose the header field, containing the corresponding values from the pop-up list on the right.

- **Uphole times** – is accessible if you use the **Calculation method: Shot Holes Using Uphole Times**

from file – the option is available, if the **Geometry and upholes from file** is on. When you choose this option the values will be read from the UT column of the chosen file.

from header - when you select this option you have to choose the header field, containing the corresponding values from the pop-up list on the right.

- **Receiver elevations** – surface altitude in receiver point

from file – this option is available, if the **Geometry and upholes from file** is on. When you choose this option the values will be read from the RE column of the chosen file.

from header - when you select this option you have to choose the header field, containing the corresponding values from the pop-up list on the right.

- **Final datum elevation** –altitude (in meters) of datum, to which the traces are reduced by means of static corrections.

Module functioning

The static corrections for each trace are calculated using the following formulas:

- **For PP**:

- **Shot statics:**

source_statics = (f_datum-source_elevation+source_depth)/replacement_vp

If you use **Calculation method: Surface source**

source_statics = (f_datum-source_elevation)/replacement_vp

If you use **Calculation method: Surface source with weathering**

source_statics = (f_datum-source_elevation+weathering_depth)/replacement_vp -

- - weathering_depth/weathering_vp

- **Receiver statics**

If you use **Calculation method: Shot holes using uphole times:**

receiver_statics=(f_datum-receiver_elevation+source_depth)/replacement_vp – uphole

If you use **Calculation method: Shot holes ignoring uphole times:**

receiver_statics=(f_datum-receiver_elevation+source_depth)/replacement_vp -

- - (receiver_elevation - source_depth)/weathering_vp

If you use **Calculation method: Surface source**:

receiver_statics=(f_datum-receiver_elevation)/replacement_vp

If you use **Calculation method: Surface source with weathering**

receiver_statics=(f_datum-receiver_elevation+weathering_depth)/replacement_vp -

-weathering_depth/weathering_vp

- For PS

Shot statics

source_statics = (f_datum-source_elevation+source_depth)/replacement_vp

If you use **Calculation method: Surface source**

source_statics = (f_datum-source_elevation)/replacement_vp

If you use **Calculation method: Surface Source with Weathering**

source_statics = (f_datum-source_elevation+weathering_depth)/replacement_vp -

- weathering_depth/weathering_vp

Receiver statics

If you use **Calculation method: Shot holes using uphole times:**

receiver_statics=(f_datum-receiver_elevation+source_depth)/replacement_vs -

- uphole weathering_vp/weathering_vs*

If you use **Calculation method: Shot holes ignoring uphole times:**

receiver_statics=(f_datum-receiver_elevation+source_depth)/replacement_vs -

-- (receiver_elevation - source_depth)/weathering_vs

If you use **Calculation method: Surface Source:**

receiver_statics=(f_datum-receiver_elevation)/replacement_vs

If you use **Calculation method: Surface Source with Weathering**

receiver_statics=(f_datum-receiver_elevation+weathering_depth)/replacement_vs -

-weathering_depth/weathering_vs

Here:

"source_elevation, source_depth, uphole – the values of surface elevation in receiver point, depth of source position in the borehole and uphole time in receiver point (they should be set for each source point). If there were no source point data on the given stake when calculating statics – the values are linearly interpolated.

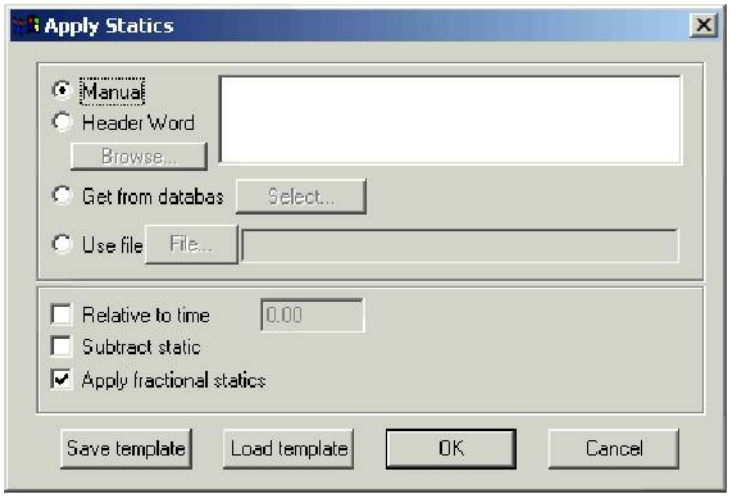
"receiver_elevation – elevation value of receiver point position (they should be set for each receiver point).

"f_datum - datum, at which the data are reduced, it doesn't change along the profile "velocities replacement_vp, replacement_vs, weathering_vp, weathering_vs are given for separate stakes and are linearly interpolated in between.

Apply Statics

This module is meant for application of static corrections which can be entered manually, read out from the database or from the file or from the trace header fields. Static shifts should be expressed in milliseconds. Negative values reduce time and shift the data upwards; positive values add the time and shift reflections deeper in section. The program interpolates corrections between specified reference points.

When this module is activated the following window appears:



There are four ways to assign correction that will be applied to traces:

- activate the **Manual** option that allows manual correction setting in the window.

An example:

CDP

0-50:500,70:300

In this example::

CDP - header field name;

0-50 - field values to which the correction is applied. In this case these are SDP points;

500 - time, etc.

i.e. here, the CDP points traces from 0 to 50 will be shifted downward by 500 ms and the CDP point trace with number 70 will be shifted downwards by 300 ms.

- activate the **Header Word** option that allows static correction setting from the header of database which can be selected in the header window that appears when clicking the **Browse...** button.

- activate the **Get from database** option that allows static corrections loading from database

object that can be selected in the standard dialog box that appears when clicking the **Select...** button.

- activate the **Use file** option that allows static corrections loading from file that can be selected in the standard dialog box that appears when clicking the **File...** button or enter the file name manually.

The text file format must be:

First line contains header for trace matching. All other lines contain two columns divided into space: header value and corresponded shift value.

Example:

TRACENO
0 50

100 60

200 75

Besides, the window contains the following options:

- **Relative to time** allows adding constant shift to static corrections that can be specified in a respective window;

- **Subtract static** option makes it possible to apply static corrections with reversed sign;

- **Use interpolation** option makes it possible to apply static corrections which value is smaller than sampling step;

Auto Statics* (Automatic static correction calculation

This module is used to calculate residual static corrections for surface conditions during land seismic observations. Static shifts are calculated relative to the “pilot” traces using mutual correlation functions (MCF) in a time window along the specified horizons and saved to the trace

header fields selected by the user. After that they can be applied to the data in a separate flow using the Apply Statics module.

This module is a so-called standalone module, i.e. it does not require any input/output modules to be present in the flow. The module reads a 2D or 3D datasets (sorted as specified by the user) and a pick set from the project database. Pilot traces are calculated along the picks in the selected time window based on a set of seismograms or a stacked section.

Static shifts are calculated relative to the “pilot” traces using mutual correlation functions (MCF) in a time window along the specified horizons. Validated shifts are then saved to the header fields specified by the user.

After that static shifts are equalized for surface conditions. The results are saved as a pick containing only the traces with permissible static shifts: a general pick (for the receiving and emitting group), a pick for the receiving group, and a pick for the emitting group.

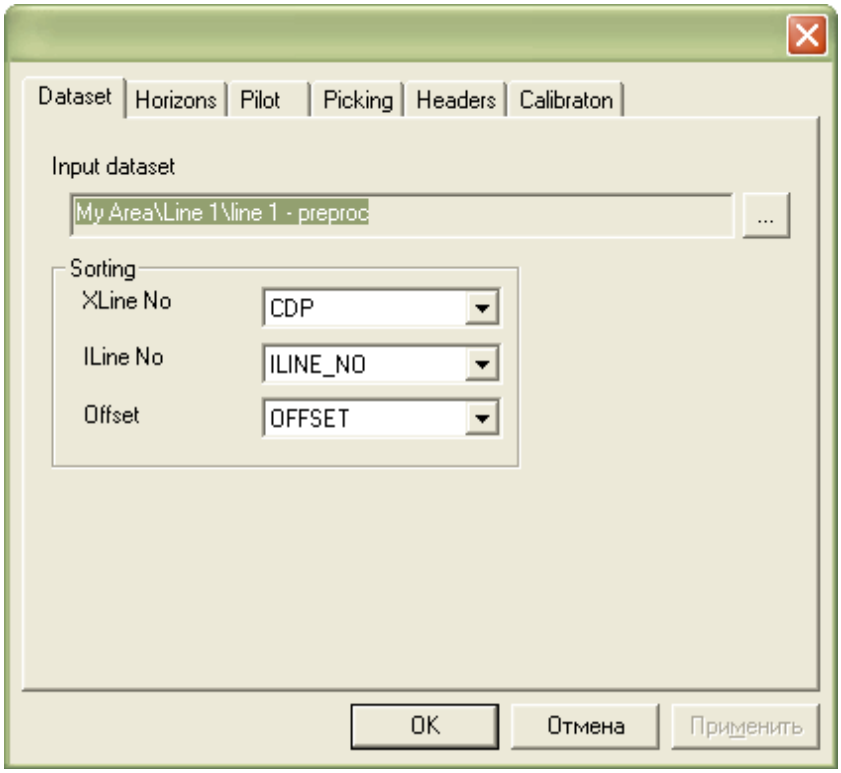
For the module to operate correctly, the input data must meet the following condition: the trace header fields TLIVE_S and TFULL_S are to be filled. The TLIVE_S field should contain the start time of the working (non-zeroed) trace area, and the TFULL_S field – the end time of the working trace area. The module will treat all trace readings falling outside the (TLIVE_ S – TFULL_S) interval as equal to zero regardless of their actual value. These headers allow limiting the time range for each trace that will be used to build the pilot trace without expressly invoking the bottom and top muting procedures. If no additional time limit is required, the TLIVE_S should be set equal to 0, and the TFULL_S – equal to the time range of the traces (for example, by using the Trace Header Math module and specifying the following two lines in the parameters: “TLIVE_S = 0” and “TFULL_S = ([NUMSMP]-1)*[dt]”).

Module parameters

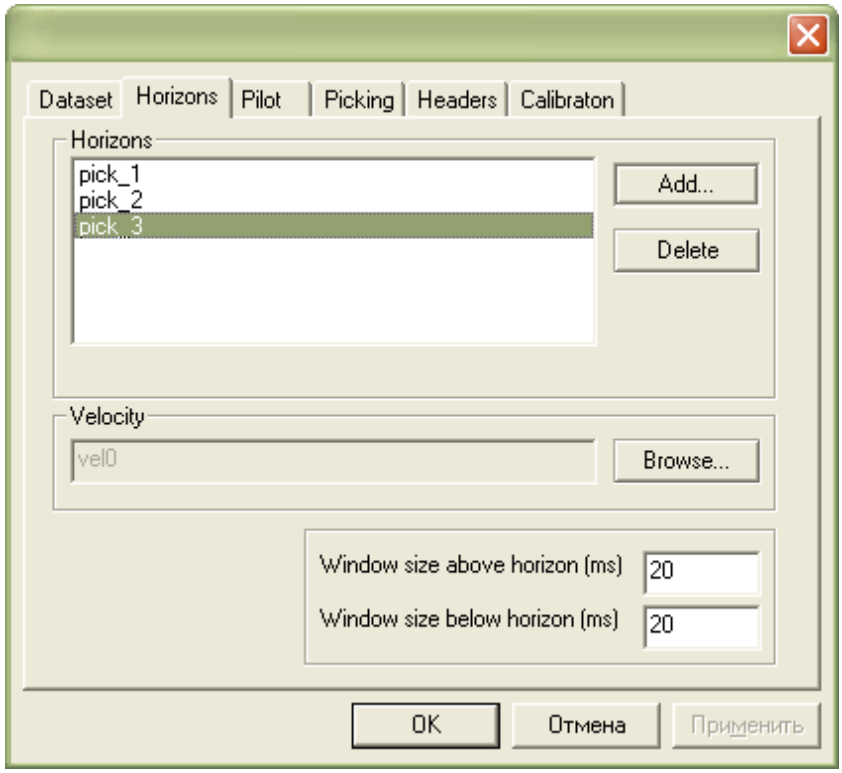
The module setup dialog box contains several tabs:

- **Dataset (Dataset selection for calculation of corrections)**

This is where you specify the path to the dataset for which the static corrections are to be calculated (**Input dataset**). The **Sorting** field is used to specify data sorting keys – such as CDP, CDP and Offset fields for 2D land survey or XLine_No, Iline_No and Offset (cross line number, inline number and offset) header fields for 3D land survey.



- **Horizons (Specification of horizons)**



The list of horizon picks along which the static shifts will be evaluated is displayed in the **Horizons** field. The picks need to be created beforehand – for example, using the Screen Display module.

It is important that the picks are tied to the same header fields as the ones specified in the first two sorting keys in the **Sorting** field on the **Dataset** tab.

You can add or delete picks from the horizon list by pressing the **Add...** or **Delete...** button, respectively.

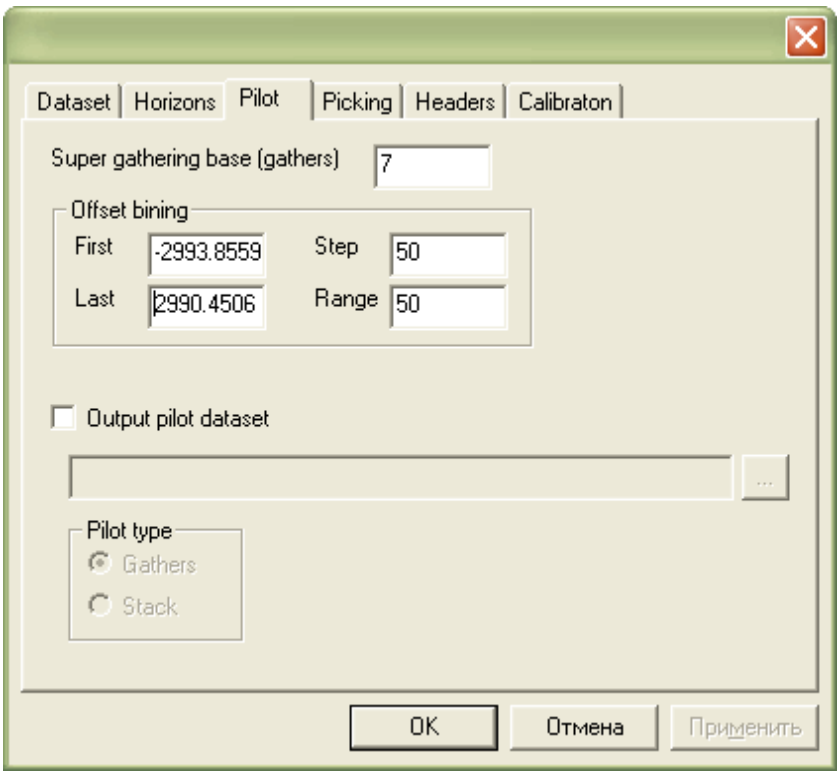
The **Velocity** field is used to specify a velocity function obtained earlier in other processing flows during vertical velocity analysis (this is the output velocity function of the Interactive velocity analysis module). Stacking velocities are used to account for the kinematic shift of traces relative to the pilot trace set before calculating residual static adjustments.

Window size above horizon (ms) – size of the window for calculation of mutual correlation functions above the horizons in ms.

Window size below horizon (ms) – size of the window for calculation of mutual correlation functions below the horizons in ms.

- **Pilot (Pilot trace set calculation)**

Parameters for pilot trace set calculation are specified on this tab.



Super gathering base (gathers)

This field is used to specify the number of ensembles that will make up the superseismogram. This parameter influences the number of traces that will be used to generate the pilot trace, but does not affect the number of pilot traces created as a result of running the module. A superseismogram consisting of n ensembles (where n is equal to the **Super gathering base** parameter value) is created for each input data ensemble.

The ensemble for which the superseismogram is created is the central ensemble in the superseismogram (except for "outer" ensembles that have less than n/2 ensembles between them and the edges of the area available to the module). For outer ensembles, the superseismogram "window" is moved in such a manner as to match the "window" edge with the available data boundary. For example, the superseismogram for the very first ensemble will consist of the first n ensembles, while such superseismogram for the last ensemble will consist of the last n ensembles.

Superseismograms may consist of several CMPs (for 2D surveys) or several In Lines or Cross Lines (for 3D surveys). The header field to be used for superseismogram generation is specified in the first sorting key in the **Sorting** field on the **Dataset** tab.

Offset binning

This group of parameters controls offset binning. It is used by the program only if the **Output pilot dataset** option is checked and the **Pilot type: Gathers** is selected.

If **Data type: Stack** is selected, all parameters from this group are ignored.

First – first offset value in meters

Last – last offset value in meters

Step – distance between offset bins in meters

Range – range of offsets falling within one offset bin.

Therefore, the number of bins will be calculated according to the following formula: (First – Last)/Step. The offset range corresponding to bin number i is an interval calculated using the following formula: (First + i*Step – Range/2, First + i*Step + Range/2).

If the **Output pilot dataset** option is enabled, the pilot trace output dataset selection button (**...**) and the pilot type radio buttons become avaliable:

Pilot type

Gathers - seismograms

Stack – stacked section

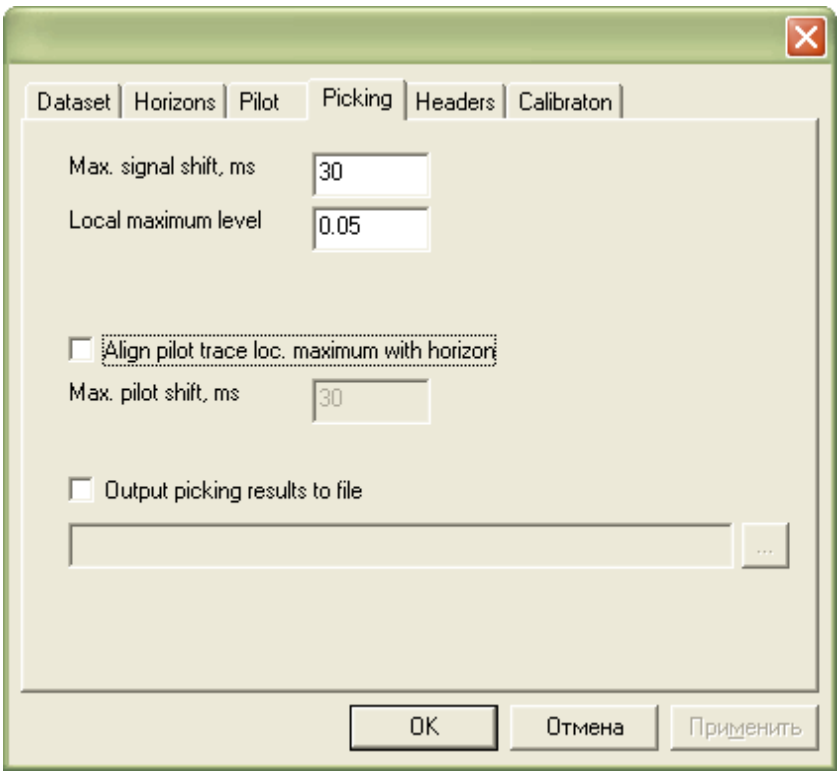
- **Picking**

This tab contains parameters used in calculation of static shifts relative to the pilot trace set. They specify a set of threshold values for calculation of permissible pilot trace shifts, shifts of the original trace relative to the pilot trace and their validation criteria, and the path to the database file in which the shifts calculated for each trace will be stored.

When calculating shifts, the maximum permissible shift to be taken into account in calculation of

static adjustments is specified, and the MCF of the original and the pilot trace is calculated. If the original and the pilot trace were identical, the MCF maximum would be achieved with a zero shift value. In all other cases within the specified time window the MCF will have one absolute maximum (A) and a number of local maximums taking different values in the vicinity of zero. Shift calculation involves finding the local MCF maximum closest to zero but above a certain threshold value calculated as p*A, where p is a user-specified parameter.

Pilot traces are built based on user-created picks. Picked values in some traces may deviate from the true horizon time. Therefore, the resulting pilot traces may be shifted relative to the picks. To eliminate this shift, pilot traces may be moved to match the position of the nearest local pilot maximum with the horizon time. The nearest local maximum is determined relative to the p*A threshold value (same as in the case described above), and the permissible shift value is determined within the limit specified by the user.



Max. signal shift, ms – maximum permissible lineup shift between adjacent traces to be taken into account in calculation of static corrections. If the calculated pilot trace shift exceeds the **Max etalon shift** parameter value, the pilot trace shift is considered invalid.

Local maximum level – p value that sets the trace and pilot MCF threshold value. It is used to find the position that is closest to zero but above the local MCF maximum threshold value when calculating the static shift.

Align pilot trace loc. maximum with horizon

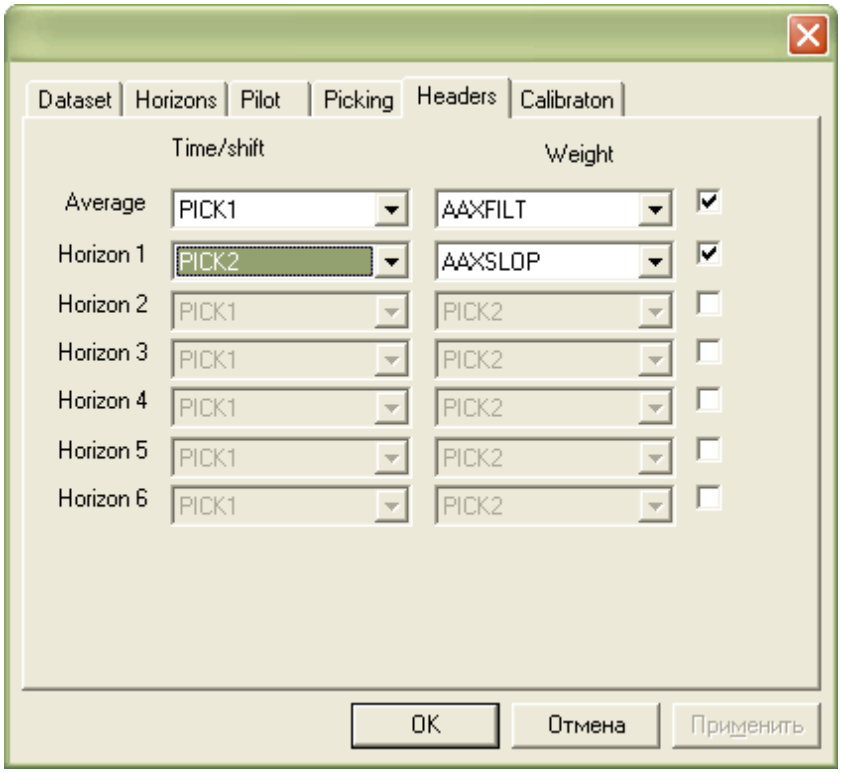
Enables the pilot shift mode. When this mode is enabled, pilot traces will be shifted to match the position of the nearest local pilot maximum with the horizon time before calculating static shifts for the specified time window. When the nearest local maximum is found, all maximums with amplitudes less then p*A will be ignored. If the calculated pilot trace shift exceeds the **Max signal shift** parameter value, no pilot trace shift will take place.

Output picking results to file

Defines the filename under which the calculated static shifts will be saved. This parameter is optional and is intended primarily for module operation analysis and troubleshooting purposes.

During normal operation the shifts are saved to the output trace headers.

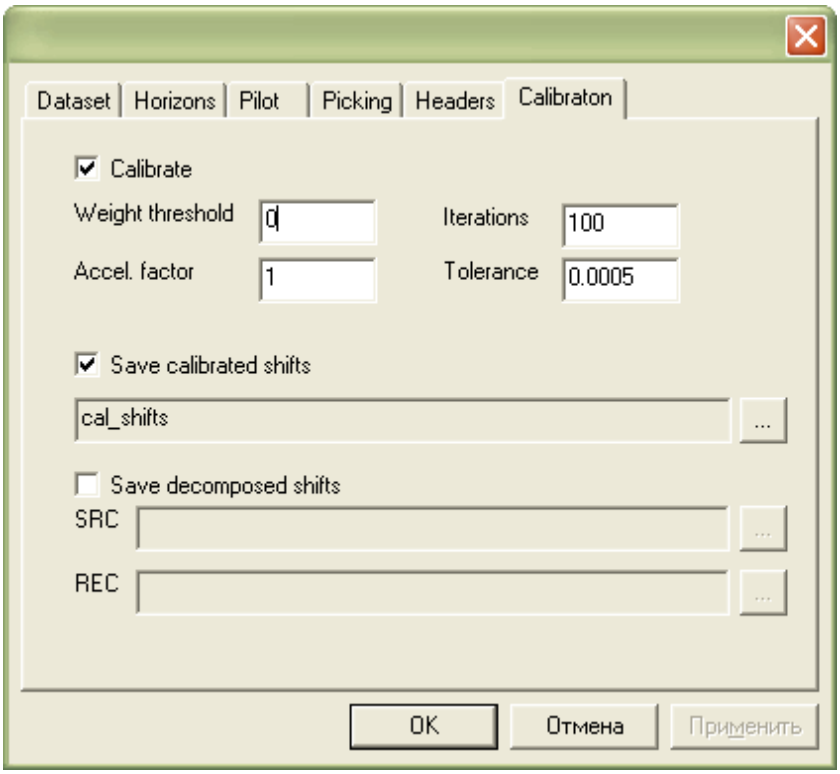
- **Headers**



This group of parameters allows specifying the names of the headers where the calculated shifts and weights for each of the horizons (**Horizon 1**, … lines) as well as the resulting average weighted shift (the **Average** line) will be saved. The sum of the calculated shift and the horizon time for the particular trace is saved to the **Time/shift** field for each horizon (time window). The checkboxes to the right of (**Average, Horizon 1**, …) lines allows selecting which values are to be output to the dataset trace headers at the module output. The **Time/shift** column is used to specify the target header fields for time shifts, the **Weight** column – the target header fields for weights used in shift averaging.

Calibration

This tab is used to set up the surface-consistent shift calculation parameters.



Calibrate – this option enables/disables the interactive surface-consistent shift calculation mode.

Weight threshold – threshold certainty value for the resulting static shifts used to reject static shifts in the 0 to 1 range. If the calculated certainty value for any particular trace is below this threshold, the shift for that trace is rejected and is not used in the procedure of data correlation for surface conditions. If this value is set to 0, all shifts are considered valid and are used in the surface consistent procedure; if set to 1, all shifts are rejected.

Accel. factor – this parameter determines the iterative process acceleration. The lowest process speed corresponds to the value of 1. Higher values will result in higher process speeds. However, increasing the speed degrades the quality of the procedure because it uses fewer traces to determine surface-consistent shifts.

Iterations – maximum number of process iterations.

Tolerance – mistie value that will result in the iterative process being terminated before the maximum specified number of iterations is completed.

Save calibrated shifts – if enabled, this option allows specifying the path for saving the calculated surface-consistent shifts to the project database.

Save decomposed shifts – save/do not save shifts associated with sources (SRC) and receivers (REC) to separate user-specified paths.

Picking (It is old module. We recommend you to use Statics Correction module)

Statics Correction.

Purpose

The module is used to calculated traces' static shifts relative to "reference" (pilot) traces on the basis of cross correlation function. Static shifts, acquired in such a way, are used for a further processing as correlation statics (trim statics) or as raw information for calculation automatic statics (auto statics). In several cases the given procedure can be used in order to adjust events' correlation (picking).

For information on generating pilot traces, see module **Pilot** description.

Application

The input data to the module Picking:

- data from the flow, containing raw data (for which the shifts are calculated) and pilot traces. The input dataset should contain one pilot trace for each ensemble (gather);

- set of horizons (picks), specifying the time intervals where cross correlation function and shifts will be calculated;

- stacking velocities, used for reflected waves' traveltime curves calculation (static shift calculation is done after considering NMO shift of traces in relation to the pilot).

Calculation of static shifts is carried out by the module in the following succession.

- Time gate boundaries are determined for each input data ensembles on the basis of the specified horizons (picks) and time window length. Further on the steps 2, 3 are carried out independently for each time window.

- Optionally (if the corresponding mode is turned on in module's parameters) traces' shift of the pilot can be done to make the position of the local pilot maximum (the closest to the horizon) coincide with the horizon time value.

- Cross correlation function of the pilot trace and each trace in the ensemble is calculated for each ensemble (seismic gather). The position of the closest local maximum of the cross correlation function determines static shift (for the given time window).

- The resulting weight average shift of the trace in relation to the pilot is calculated for each trace of raw data. Averaging is done between the shifts, acquired for given trace independently for each time window. The weight of a specific shift depends on the value of the local extremum of cross correlation function of the pilot and a trace and equals to the ratio of the local maximum and the value of the absolute maximum (p1). If the mode of reference shift is selected, the weight depends as well on the value of the local reference maximum, which position corresponds to the horizon position after the shift. "Reference weight" (p2) equals in this case to the ratio of the local and the absolute maximums, while the final weight, with which the shift is averaged, equals to geometrical mean of p1 and p2 (SQRT(p1*p2)).

Depending on the mode of operation a set of pilot traces or a set of raw traces (for which the shifts were calculated) is output from the module. The raw traces' output mode is the main operational mode. The indicated headers of the raw traces set contain a resultant static shift, as well as shifts, acquired separately on different time gates (more precisely, the sum of shift and reference horizon is output, for a more convenient visual control of the obtained correlation shifts in the form of picks or headers output on the screen (in time domain) in Screen Display module), and weights, which were used for averaging. The mode of pilot traces output is very convenient for validity check of input data to the module and validity check of the reference shift (if the reference shift is applied).

Very often (especially when we do not use the mode of reference shift) we don't need a detailed picking of reference horizons inn order to obtain correlation shifts, we can confine ourselves to rather rough (smoothed) horizon picking on the section.

Time marks of reference horizons have influence on:

- time gates' boundaries;
- value of reference trace shift, if the mode of reference trace shift is on.
- time of "picking", output in the output dataset in headers, for each horizon (they have no direct influence on the resultant weighted average time shift of the reference horizons). The output value for each time gate equals to the sum of the time mark of the reference horizon and the calculated shift.

The time marks of the reference horizon have no direct impact on static shift value, calculated as a result of correlation with the reference.

Input data characteristics

The data input to the module in the flow should comply with the following conditions for the correct module performance:

- Input data should contain pilot traces, and traces for which the time shifts will be calculated.
- Pilot traces should contain the following header field TRC_TYPE, equal to 2. Raw traces (subject to shift) should have TRC_TYPE, equal to 1.
- Each ensemble of the input data should consist of one pilot trace and one or more traces, for which the shifts will be calculated. The pilot trace should be in the first ensemble.

Example 1.

We need to calculate static shifts for the traces relative to the pilot, generated for CDP gathers. Suppose, that the dataset is prepared that contains pilot, in which one trace corresponds to each CDP, header field TRC_TYPE is equal to 2 for all traces. Raw (data being shifted) are located in another dataset, containing one seismic gather for each CDP, header field TRC_TYPE is equal to 1 for every trace. Thus, we set the following parameters in the module Trace Input:

- in Datasets list – datasets of pilot and raw traces;

- in Sort fields – CDP, TRC_TYPE (strictly in the indicated sequence);

- in Selection field - the line "*:2,1"; -

We get the needed sorting order. Ensemble key in the given example is considered to be the CDP number, the first trace of each output ensemble of the module Trace Input will contain the pilot trace, the rest traces are considered to be CDP gather of raw data. Module Picking will calculate a shift for each trace of raw data with reference to the pilot trace, independently for each ensemble.

Example 2.

We need to calculate static shifts for each trace of the profile relating to the single pilot trace in case of single channel profiling. Pilot trace is located in a separate dataset, TRC_TYPE is equal to 2 for the pilot trace, and 1 for every trace of the profile; Header field of the trace XLINE_NO contains one and the same profile number (for example, 1) in both datasets.

We have to set the following parameters in Trace Input:

- in Datasets list – pilot dataset and profile dataset;

- in Sort fields – XLINE_NO, TRC_TYPE, CDP (strictly in the indicated sequence). Instead of CDP quite often there can be other header field, depending on the field which contains the number of profile point. In the given example the presence of the third header sort field is not mandatory, if we don't need to guarantee the trace output order after calculating the shifts.

- in field Selection – the line "*:2, 1: *"

With this way of parameters' selection, all the dataset read (joined from the two datasets within the database), will represent one ensemble (the first sort key is the same for all traces); the first trace will be the pilot trace. Static shifts will be calculated for all traces of the profile.

Example 3.

We need to calculate static shifts of each trace of the profile related to its own pilot trace (acquired for example, by means of sliding averaging of raw profile along the reference horizon, or using the module Pilot) in case of single channel profiling. Let's assume, that a dataset containing a pilot is prepared, which contains one trace per each profile point, the number of profile point is given by CDP field, the header field TRC_TYPE equals to 2 for every trace. The raw data (being shifted) is located in another dataset, the number of a profile point is given by CDP field, TRC_TYPE field is equal to 1 for all traces.

You should specify the following parameters in Trace Input:

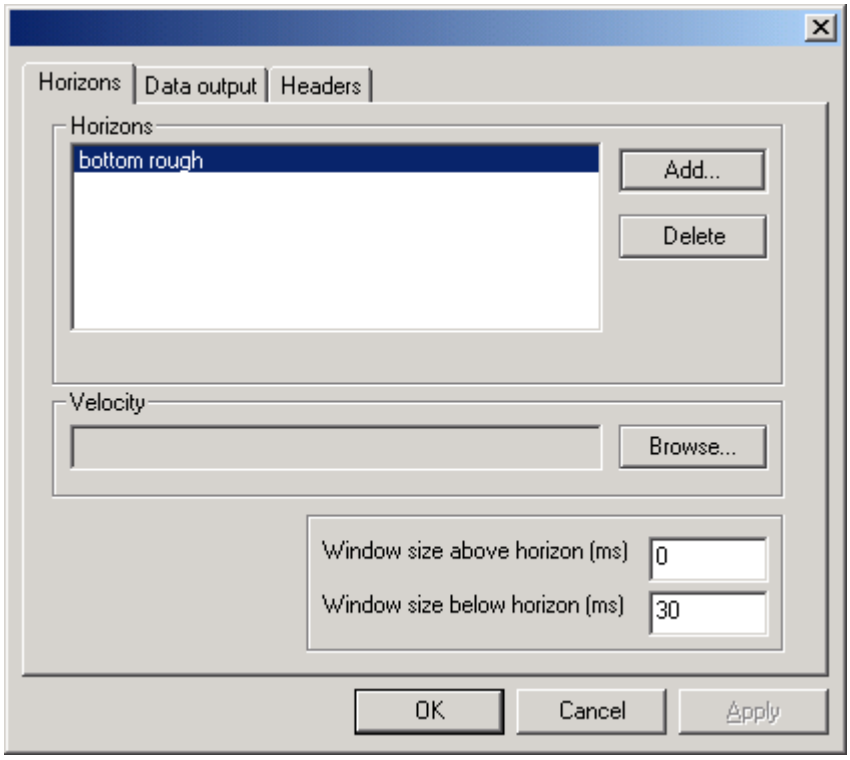
- in Datasets list – datasets of pilot and the data;
- in Sort fields – CDP, TRC_TYPE (strictly in the indicated sequence).
- in Selection field – the line "*:2, 1"

With this way of parameters' selection, each ensemble will contain two traces – pilot trace (the first one), and data trace. A static shift will be calculated n regard to the pilot trace for each data trace in Picking module.

- Input data sorting should correspond to pickings match fields, which determine time windows of the pilot. For example if the match key of pickings is CDP, the first sort key of the input data should be CDP.

Parameters

Group of parameters Horizons



- **Horizons**

The list represents horizons, which determine the set of time windows. A number of time windows, which will be used is equal to the number of horizons in the list. Each time window for the given pilot trace is determined as time interval from t0-t1 to t0+t2, where t0 – time mark of the corresponding pick of the given trace, t1 – value of Window size above horizon parameter (see below), t2 – value of Window size below horizon parameter (see below).

If the picking point is absent for the given trace, t0 is acquired by simple linear interpolation between the neighboring picking points.

Using Add… button to the right of the list you can add a horizon from the system database. Delete button to the right of the list removes the selected horizon from the list.

NOTICE: pickings' match fields should coincide for all picks selected to the list. The input data sorting should correspond to pickings' match fields. For instance if the match field of the pickings is CDP, the first sort key of the input data should be CDP.

- **Window size above horizon (ms), Window size below horizon (ms)**

These parameters correspond to the currently selected horizon from the Horizons list, and specify the boundary of time window (see description **Horizons**).

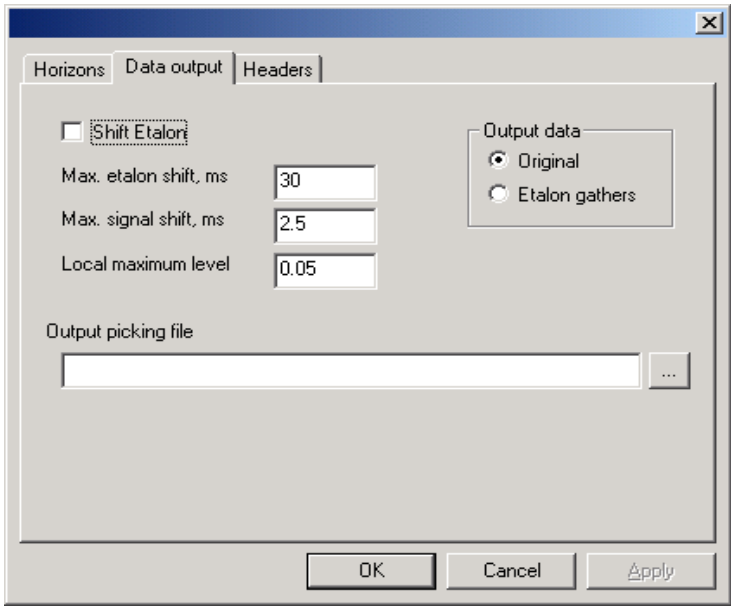
- **Velocity**

The Browse… button to the right of parameter allows selecting velocity model from the database.

Specification of stacking velocities allows considering NMO shift for correct calculation of static shift. Stack velocities are used for calculation of time values on raw traces (for which the shifts are calculated), with reference to which the shift is calculated. Tm(t0, Velocity) is subtracted from the time value estimated from the maximum of cross correlation function in order to calculate the final shift for the specified time window, where Tm – hyperbolic traveltime curve function of reflected wave, t0 – the given time mark on pilot trace, velocity – stacking velocity for CDP point, to which the trace is related, and time t0.

This parameter is not mandatory. In case this parameter is absent, Tm(t0, Velocity) is set to zero. There is no need to assign velocity model, if the data input to the module are the data with applied NMO correction, either stack (or single channel data with zero offset). In the first case (applied NMO corrections) setting stack velocities will lead to incorrect stacking of pilot data.

Group of parameters Data Output



- **Output data**

Specifies the mode of module operation.

In Original mode the raw traces for which the shifts are calculated and saved to the given header fields are output from the module (pilot traces are not output). The given mode of the module operation is considered to be the main operation mode.

In Etalon gathers mode the pilot traces are output from the module. The given mode is convenient for the validity check of the input data definition and visual control of the reference shift results (if the reference shift mode was activated).

- **Shift etalon**

Activates the mode of reference shift. If the mode is activated, the pilot traces will be shifted to allow the position of the neighboring local maximum coincide with horizon time mark before calculating static shifts for the given time window. In addition, the maximums with the amplitude values less than p*A, where A – the value of the absolute maximum of the reference within the time window, p – the value of parameter Local maximum level, are ignored when calculating the neighboring local maximum.

If the calculated pilot trace shift exceeds the value of parameter Max etalon shift, there will be no pilot trace shift.

- **Max. etalon shift, ms**

Is used only in case when the reference shift mode is activated. If the calculated pilot trace shift exceeds the parameter Max etalon shift value, the pilot trace shift is not performed.

- **Local maximum level**

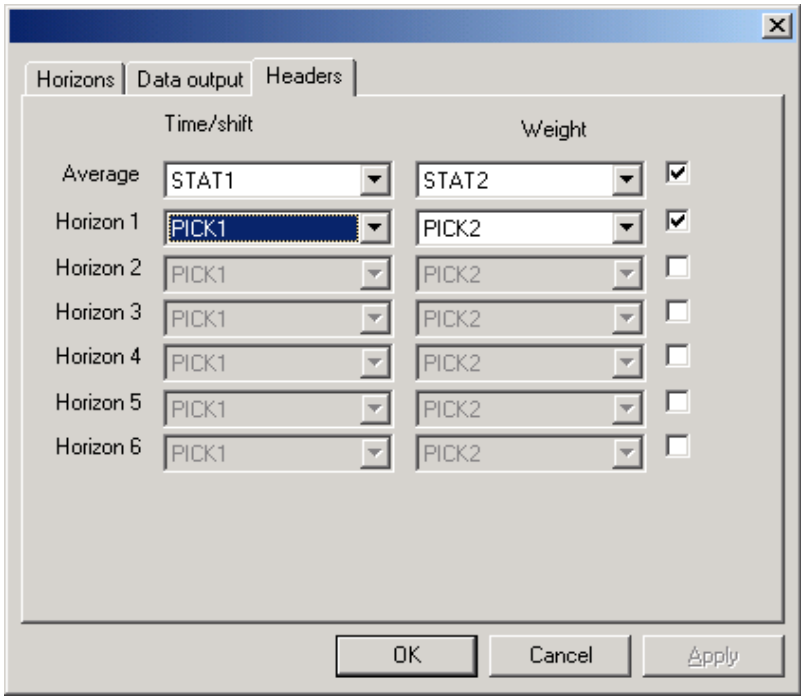
For the calculation of the trace shift relative to the pilot the module Picking finds the position of the neighboring local maximum of the trace and the pilot cross correlation function, exceeding p*A in modulus, where A – the value of the absolute maximum of the cross correlation function within the

time window, p – the value of parameter Local maximum level.

- **Output picking file**

Specifies the filename, to which the calculated static shifts will be output. The given parameter is not mandatory; the shifts are saved to the output traces' headers in operation mode, the given possibility is provided in order for module parameters' debugging.

Group of parameters Headers



The given group of parameters allows specifying header names, to which the calculated shifts and the weights for the given horizon will be saved (lines Horizon 1, …), as well as the resultant weight-averaged shift (line Average)

In addition for each horizon (time window), a sum of the calculated shift and horizon time mark will be written to the field Time/shift for the given trace.

A flag to the right of the line (Average, Horizon 1, …) allows selecting the values output to the trace headers of the dataset at the output of the module.

The column Time/shift determines header fields, where the time shifts are output; the header fields for weight output, which were used during shifts' averaging, are set in the column Weight.

Auto Statics* (Calculation of automatic statics)

The module is intended for calculation of residual statics for the surface conditions with terrestrial seismic observations. Static shifts are calculated in relation to "pilot" traces on the basis of mutual correlation functions (MCF) in a time window along the assigned horizons and are saved in the trace header fields selected by the user. Further on, they can be inserted into the data in a separate flow by means of Apply Statics module.

This module belongs to the class of independent (Stand-alone) modules, i.e. it does not require any other I/O modules in the flow. The module reads a set of 2D or 3D data from the project database in the sorting defined by the user, and a set of pickings. Pilot traces are calculated in the assigned time window along pickings on a set of gathers or using the total section.

Static shifts are calculated in relation to "pilot" traces on the basis of mutual correlation functions (MCF) in a time window along the assigned horizons. The shifts recognized as reliable are then saved in the trace header fields selected by the user.

Further, there is an alignment of static shifts for the surface conditions. The result is saved in the form of picking where only traces with admissible static shifts are involved: general picking (for the receiving-emitting group), picking for the receiving group, picking for the emitting group.

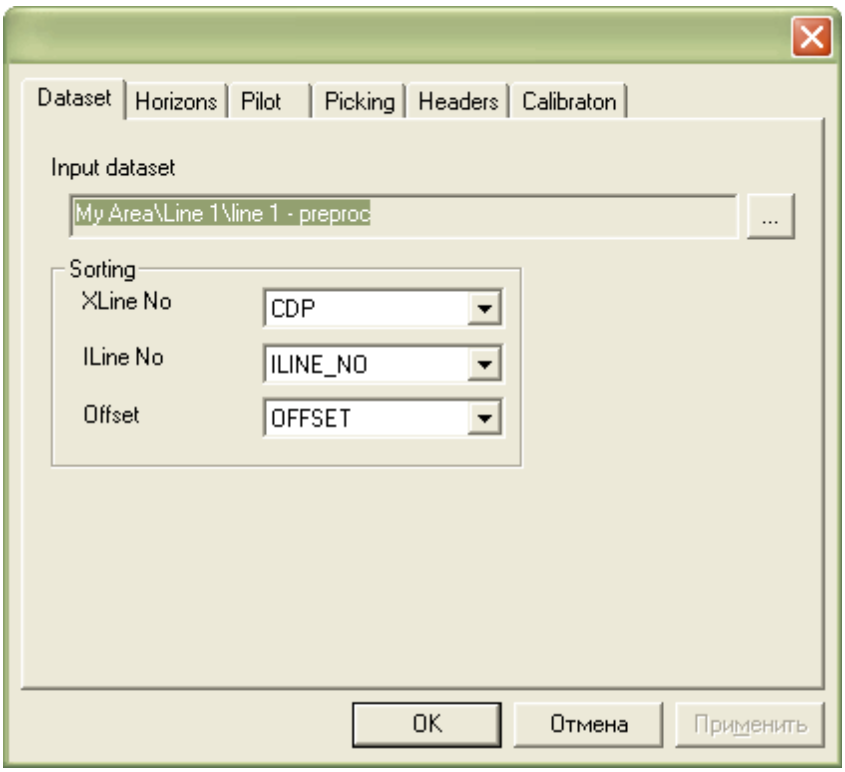
For proper operation of the module, the data at the input must satisfy the following condition: the traces are to have TLIVE_S, TFULL_S header fields filled. Field TLIVE_S is to contain starting time of the operating (not resetted) trace area, TFULL_S is to contain end time of the operating trace area. Trace counts outside the interval (TLIVE_ S – TFULL_S) module will be regarded as zero, irrespective of their actual value. These headers help each trace to restrict the time span which will be used for pilot creation, without explicit application of lower and upper muting procedure. If no additional time limit is required, one is to set TLIVE_S to 0 and TFULL_S to the time range of the traces (for example, by means of Trace Header Math module by setting two lines in parameters: “TLIVE_S = 0” and “TFULL_S = ([NUMSMP]-1) * [dt]”).

Module Parameters

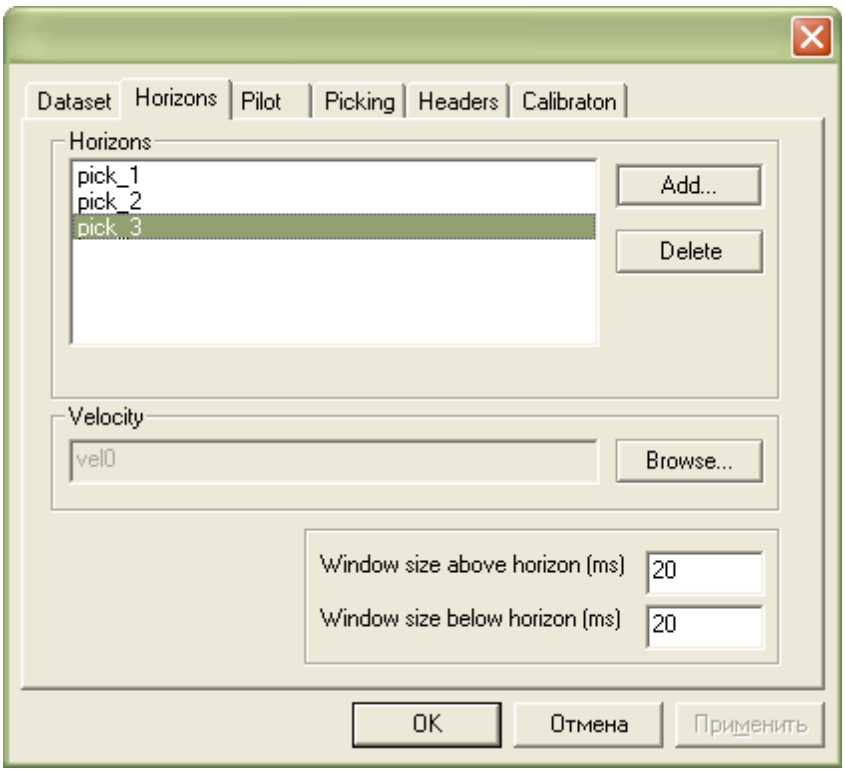
Module parameter dialogue contains several tabs:

- **Dataset (selection of dataset for statics calculation)**

This tab shows the path to the dataset for which statics will be calculated (**Input dataset**). In the **Sorting** field data sorting keys are set. For example, for 2-dimensional terrestrial survey CDP, CDP, Offset fields can be used, while for 3-dimensional terrestrial survey XLine_No, Iline_No, Offset (inline number, crossline number, offsetting) can be applied.



Horizons (Horizons assignment)



In the **Horizons** field the list of horizons pickings along which static shifts will be estimated is displayed. The pickings are to be created in advance, for example, in Screen Display module.

It is important for pickings to be anchored to the same header fields specified in the first two sort

keys in the **Sorting** field in **Dataset** tab.

One can add or delete pickings from the list of horizons by means of buttons **Add...** or **Delete...** accordingly.

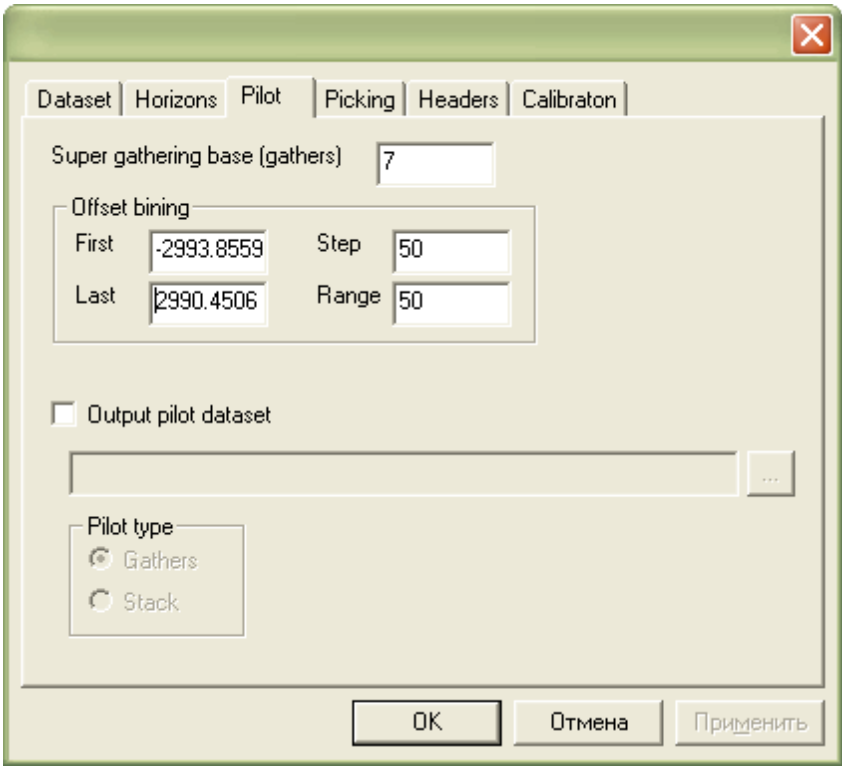
In the **Velocity** field the velocity function acquired earlier in other processing flows at the vertical velocity analysis is set (being the output velocity function of Interactive velocity analysis module). Stacking velocities are used for registration of kinematic shift of traces in relation to the pilot set of traces prior to calculation of residual statics.

Window size above horizon (ms) is the size of the window for mutual correlation functions calculation above horizons, in ms.

Window size below horizon (ms) is the size of a window for mutual correlation functions calculation below horizons, in ms.

Pilot (Calculation of the pilot set of traces)

This tab shows parameters for calculation of the pilot set of traces.



Super gathering base (gathers)

The number of assemblies which will compose the super gathering base is assigned in the field. This parameter sets the amount of traces which will be used for acquisition of the pilot trace. For each assembly of input data a super gathering base consisting of n assemblies is formed (n equals to the value of **Super gathering base** parameter).

The assembly for which the super gathering base is formed is the central assembly in the super gathering base (apart from “extreme” assemblies which fall back less than n/2 assemblies from the edges of the data area accessible to the module). For extreme assemblies the super gathering base "window" moves in the manner so that the window edge coincided with the limit of the accessible data. For example, for the very first assembly the super gathering base will consist of first n assemblies, for the last one – of last n assemblies.

Super gathering bases can consist of several CDPs (for 2D) or several In Lines, Cross Lines (for 3D). The header field on which super gathering bases are formed is set in the first sort key of the **Sorting** field of **Dataset** tab.

- **Offset binning**

This group of parameters is intended for binning according the offsets. It is used by the program only in case the **Output pilot dataset** option is enabled and the type of pilot set of traces **Pilot type: Gathers** is selected.

If **Data type: Stack** is selected the parameters from this group are ignored.

First – value of the initial offset in meters

Last – value of the final offset in meters

Step – distance between offset-bins in meters

Range – range of offsets getting in one offset-bin.

Thus, the amount of bins will be calculated under the formula: (First – Last)/Step. The range of offsets corresponding to bin number i equals the interval calculated under the formula: (First + i*Step – Range/2, First + i*Step + Range/2).

- If **Output pilot dataset** option is enabled, the button of output dataset for pilot traces selection (**...**) and traces pilot set type selection become available:

- **Pilot type**

Gathers - seismograms

Stack – stack

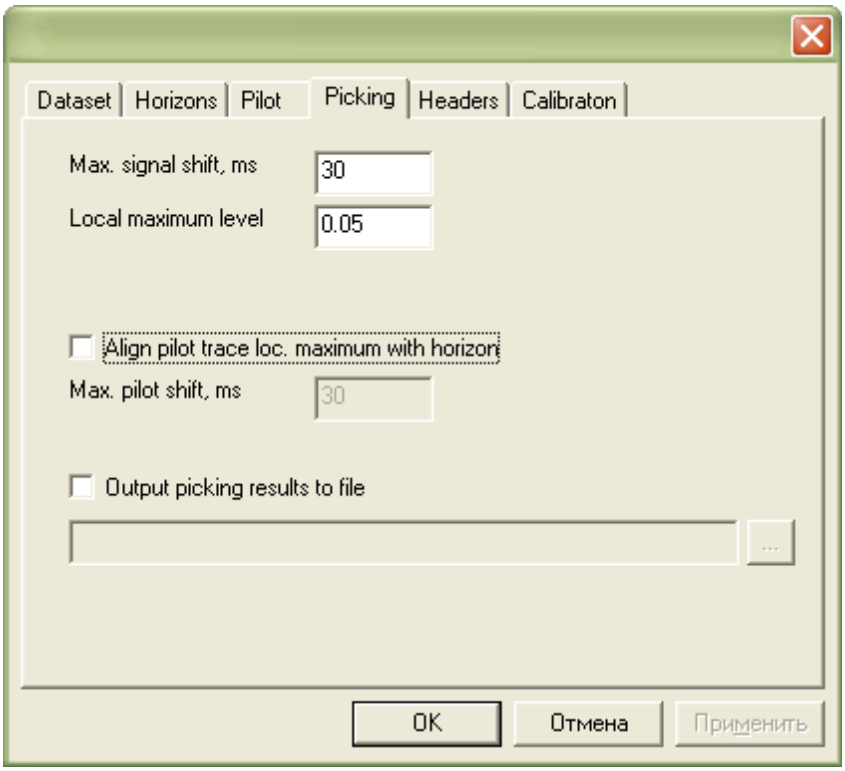
- **Picking**

Parameters of this tab are intended for calculation of static shifts in relation to the pilot set of traces. Parameters assign the set of threshold values for calculation of admissible shifts of the pilot trace, shifts of the initial trace in relation to the pilot one and criteria of their reliability, and the path to the database file in which the calculated shifts for each trace will be stored.

In the process of shifts calculation the maximum admissible shift that will be considered at calculation of statics is assigned and MCF of the initial and pilot traces is calculated. If the initial and pilot traces coincide, the maximum MCF can be reached at zero value of the shift. Otherwise, within the set time window MCF will have one absolute maximum (A) and numerous local maxima

of various values in proximity to the zero value. When calculating the shift, the nearest to zero local MCF maximum exceeding some threshold value calculated as p*A, where p is the parameter set by the user, is searched.

When designing the pilot traces, the pickings created by the user are used. Picked values on particular traces can deviate from the true time for horizons. Therefore, the acquired pilot traces can have shifts in relation to pickings. For elimination of this shift, the pilot traces can be moved so that the position of the nearest local pilot maximum coincided with the horizon time. The nearest local maximum is defined in relation to the threshold value p*A (like in the case described above), and the value of admissible shift is defined within the value set by the user.



Max. signal shift, ms is the maximum allowed wave pattern shift between adjacent traces that will be considered at calculation of statics. If the calculated pilot trace shift exceeds the value of parameter Max signal shift, the pilot trace shift is considered unreliable.

Local maximum level is the value p that sets the MCF threshold value of the trace and the pilot for search of the position of the local maximum of MCF nearest to zero and exceeding the threshold value while calculating the static shift.

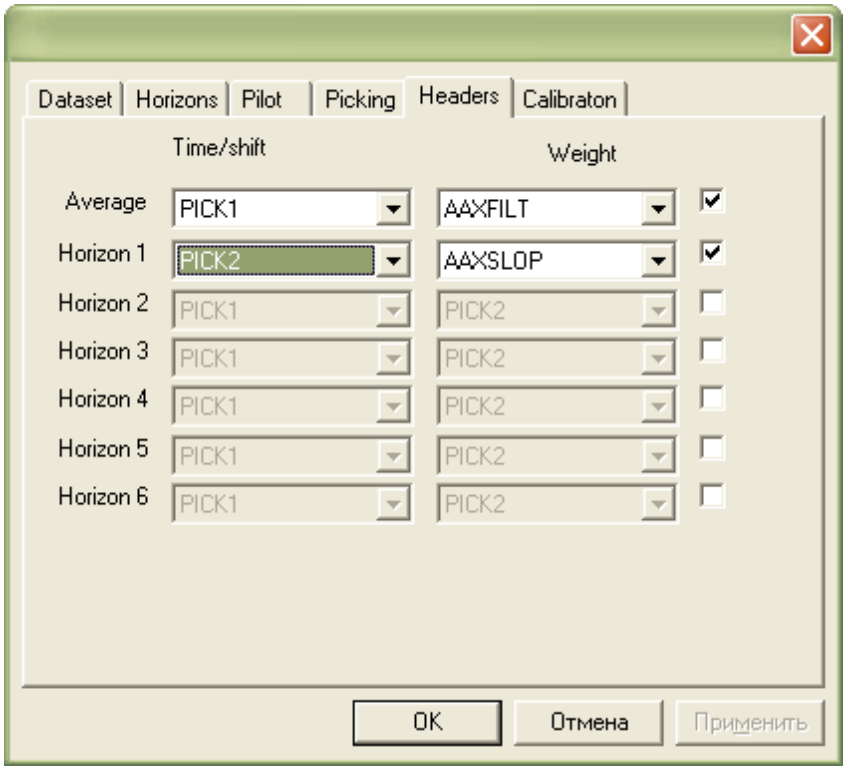
Align pilot trace loc. maximum with horizon

Includes the pilot shift mode. If the mode is enabled, prior to static shifts calculation for this time window, the pilot traces will be shifted so that the position of the nearest local pilot maximum coincided with the horizon time. Upon this, when searching for the nearest local maximum, maxima with amplitudes below p*A are ignored. If the calculated pilot trace shift exceeds value of parameter Max signal shift, the pilot trace shift is not executed.

Output picking results to file

Defines the file name where the calculated static shifts will be written. This parameter is optional and is intended mainly for the analysis of the module operation and troubleshooting. In the operating mode the shifts are saved in the output trace headers.

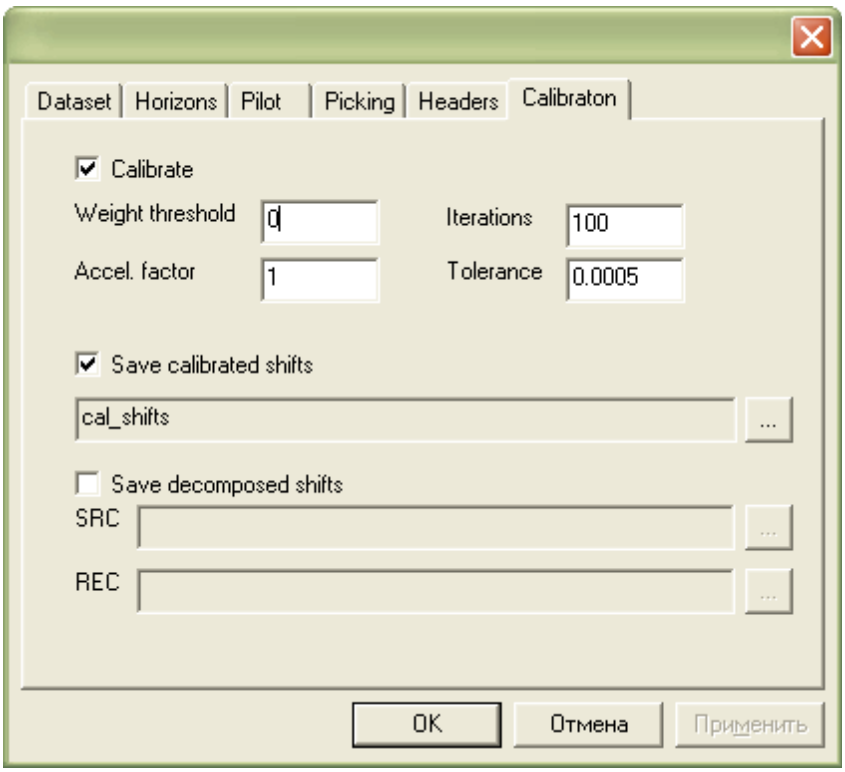
Headers



This group of parameters enables to set names to the headers where the calculated shifts and weights for each horizon will be saved **(lines Horizon 1, …),** as well as the resultant weight-average shift (line **Average**). At the same time, field **Time/shift** contains the total of the calculated shift and the horizon time for this trace for each horizon (time window). The toggle to the right of the line (**Average, Horizon 1, …**) enables selection of the values to be brought to the dataset trace headers at the module output. In column **Time/shift** header fields intended for the time shifts are set, in column **Weight** header fields for the weight outputs used at averaging of the shifts are set.

Calibration

Parameters setting tab for calculation of the surface-coordinated shifts.



- Option **Calibrate** enables/disables the iteration mode of the surface-coordinated shifts calculation.

Weight threshold is the threshold value of estimation of the acquired static shift reliability used for rejection of static shifts in the range of 0 to 1. Thus, if the calculated estimation of shift reliability for any trace is below the threshold value, the shift for this trace is rejected and is not used in the data coordination procedure for the surface conditions. If the value is 0 all shifts are considered reliable and are used at calculation of surface coordination, if the value is 1 all shifts are rejected.

Accel. factor is the parameter that defines acceleration of the iteration process. The lowest velocity of the process at this parameter value will be 1. If the parameter is higher, the velocity increases. The higher the velocity the lower is the quality of operation, as it leads to the decrease of the amount of traces participating in the surface-coordinated shifts determination procedure.

Iterations field specifies the maximum amount of iterations in a process.

Tolerance is the value of discrepancy which sets the limit when the iteration process will be terminated before completion of the assigned maximum number of iterations.

- **Save calibrated shifts** is the option specifying the path to save the calculated surface-coordinated shifts in the project database, if enabled.

- **Save decomposed shifts** saves/does not save the source-conditioned shifts (SRC) and the receiver-conditioned shifts (REC) separately in accordance with the paths specified by the user.

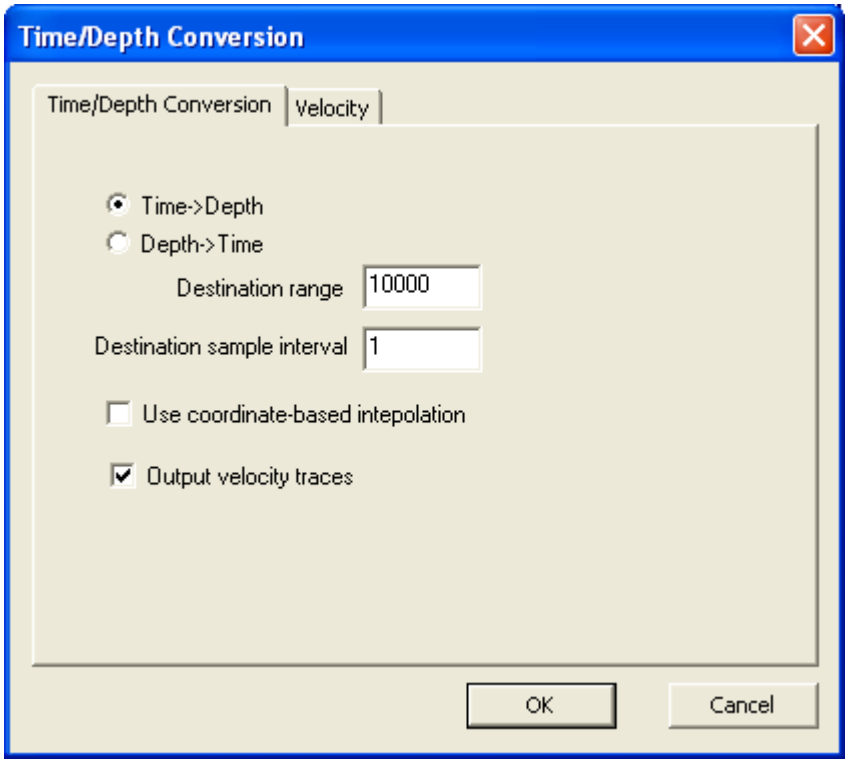
Velocity

Time/Depth Conversion

The module converts the data from time to depth domain and vice versa using a specified velocity model. Velocities can be either RMS or interval, defined in either time or depth domain. Conversion can be based on either RMS or average velocities.

The velocity model is defined at specific CDP's and is interpolated in between of them. Below the deepest point where velocity is explicitly defined, the RMS/average velocities are calculated basing on the assumption that the interval velocity remain unchanged.

When the module is activated the following dialog appears:



Two tabs are available: **Time/Depth Conversion** and **Velocity**.

- **Time/Depth Conversion tab** contains the following parameters:

- **Time->Depth** and **Depth->Time** specify the direction of the conversion;

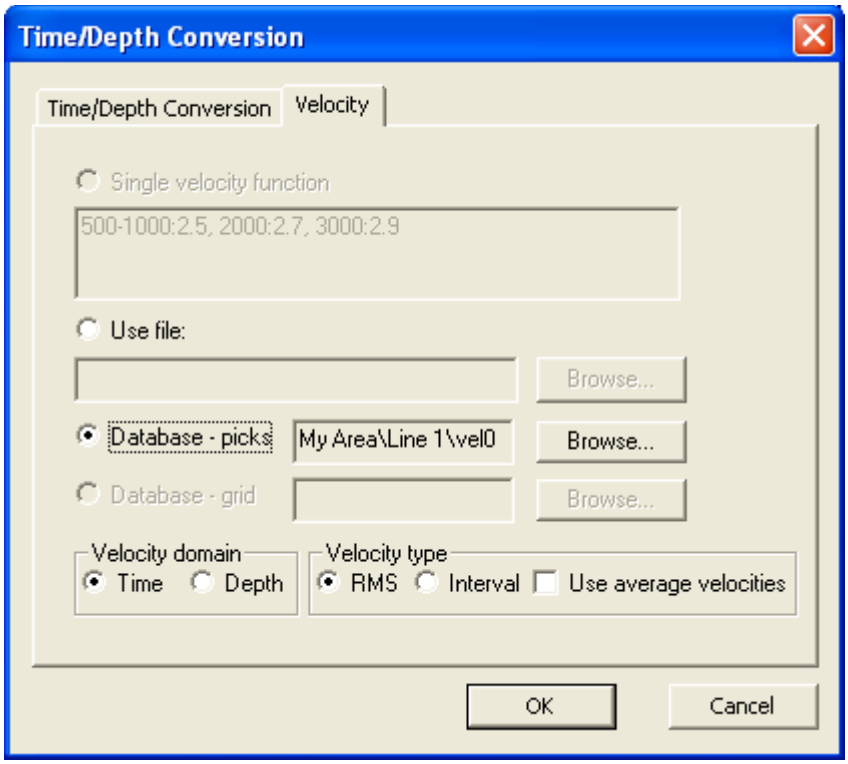
- **Destination range** – specify the maximum depth or time of the resulting traces after the conversion, either in m or in ms depending on the output domain;

- **Destination sample interval —** sample interval of the resulting traces after the conversion, either in m or in ms depending on the output domain;

- **Use coordinate-based interpolation** – as it has been mentioned, velocities are defined at certain reference CDP's. Between those reference CDP's they are linearly interpolated. When the option is off, the interpolation is based on CDP indexes. Otherwise, interpolation is made according to the CDP coordinates stored in CDP_X and CDP_Y header fields of each trace.

- **Output velocity traces** – when the option is on the resulting traces in the output domain will be substituted by the interpolated values of either RMS velocities or average velocities, whichever are to be used for the conversion depending on the **Use average velocities** flag on the next tab.

- **Velocity** tab contains the following parameters:



- **Single velocity function** – select this option to specify velocity constant on lateral in the edit window shown below. The syntax is similar to that of normal moveout correction application,

velocities are in km/s.

- **Use file** – select this option to read out the velocity function from the text file (usually created by either **Interactive Valocity Analysis** or by **Velocity Editor** and, perhaps, it is spatially heterogeneous). To select a file, click the **Browse...** button or specify the name manually.

- **Database – picks** – select this option to read the velocity function from velocity picks saved in the project database (usually created by the **Interactive Velocity Analysis**). To select a pick, click the **Browse...** button.

- **Velocity domain** – this parameter defines in what domain this velocity field is specified: **Time** or **Depth**.

- **Velocity type** – this parameter defines the type of specified velocity: **RMS** (root-meansquare) or **Interval**.

- **Use average velocities** – when this flag is unchecked, the time-to-depth (or depth-to-time) conversion is performed using RMS velocities (input velocity model, if not RMS, will be scaled to RMS velocities). Otherwise, average velocities will be used for the conversion (input velocities will be scaled to average velocities). In case, input velocities are RMS and the flag is checked, average velocities will be calculated via interval velocities.

The formulas used for velocity scaling in case of two-interval model are shown below (for N-interval model recursive approach is used):

RMS velocities from interval velocities:

$$v_{эф}^2 = \frac{v_1 * h_1 + v_2 * h_2}{\frac{h_1}{v_1} + \frac{h_2}{v_2}}$$

Here $v_{эф}$ — RMS velocity, h_1, h_2 — thicknesses of the intervals, v_1, v_2 – interval velocities.

Average velocities from interval velocities:

$$v_{cp} = \frac{h_1 + h_2}{\frac{h_1}{v_1} + \frac{h_2}{v_2}}$$

Here v_{cp} — average velocity.

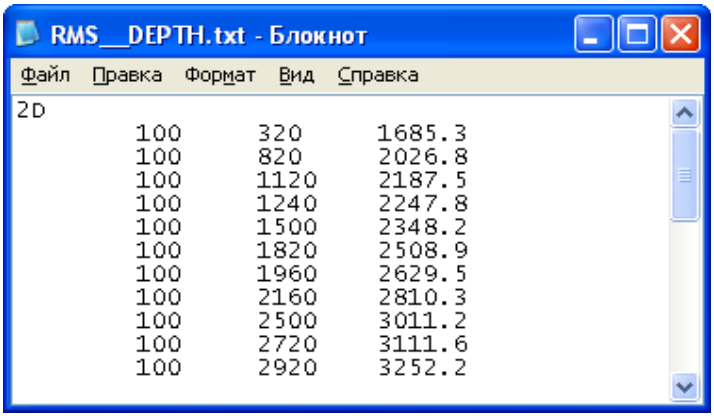
Interval velocities from RMS velocities:

$$v_2 = \sqrt{\frac{v_{эф2}^2 * t_{02} + v_{эф1}^2 * t_{01}}{t_{02} - t_{01}}}$$

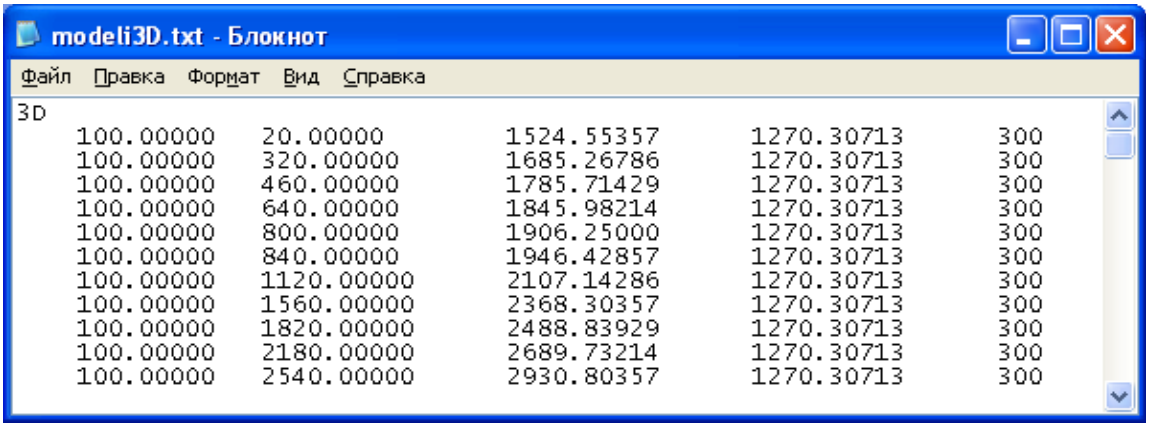
Here $v_{эф1}$ — RMS velocity at time t_{01}, equal to the interval velocity in the uppermost interval 1, $v_{эф2}$ — RMS velocity at time t_{02}, v_2 – interval velocity of the interval 2.

Appendix 1. Velocity file format.

The file is ASCII tabulated, either space or tab separated. First line contains data type: 2D or 3D. Then, there are columns containing CDP numbers, times (or depths), velocities, and (optionally) X and Y coordinates. Velocities can be either RMS measured at the specified time (depth) or interval corresponding to the interval above the specified time (depth) until the previous time (depth). The examples are below:



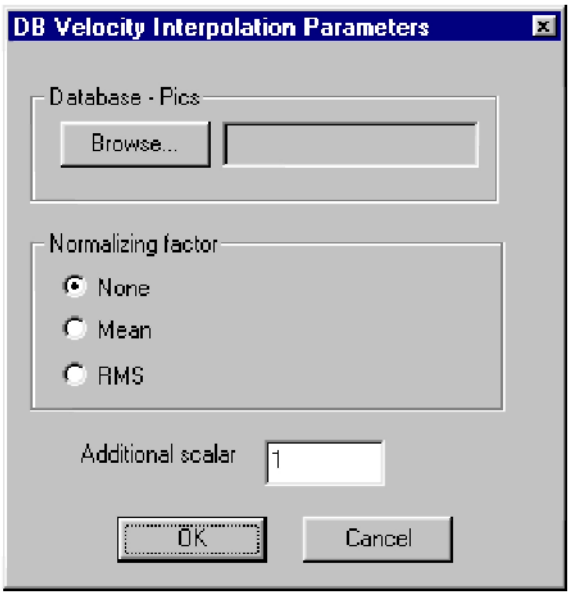
Here the model type is 2D, the columns are: CDP, time (or depth), velocity.



Here the model type is 3D, the columns are: CDP, time (or depth), velocity, X-coordinate, Y-coordinate.

DB Velocity Interpolation

This module is meant for velocity field visualization together with CDP section added to it. When this module is activated the following window appears:



In the **Database-Pics** field, click the Browse... button and, in the window that opens, select the file with required velocities from the database.

A value of the point displayed on the screen will equal the sum of velocity values in this point and trace amplitude in this point multiplied by normalizing factor.

The normalizing factor is defined in **Normalizing factor** field:

- **None** - zero normalizing factor. In the running window only velocity field will be displayed,
- **Mean** - in the running window the seismic section normalized by mean value will be added to velocity field,
- **RMS** - in the running window the seismic section normalized by root-mean-square value will be added to velocity field.

In the **Additional scalar** the additional scalar on, which the trace samples values will be multiplied before being displayed on the screen, should be specified.

Usage (An example of module usage in the flow)

A typical flow would includes:

Trace Input

DB Velocity Interpolation

Screen Display

Trace Input reads out a summed section saved in the database.

DB Velocity Interpolation - velocity field interpolation module itself.

Screen Display displays velocity field with superposed section on the screen.

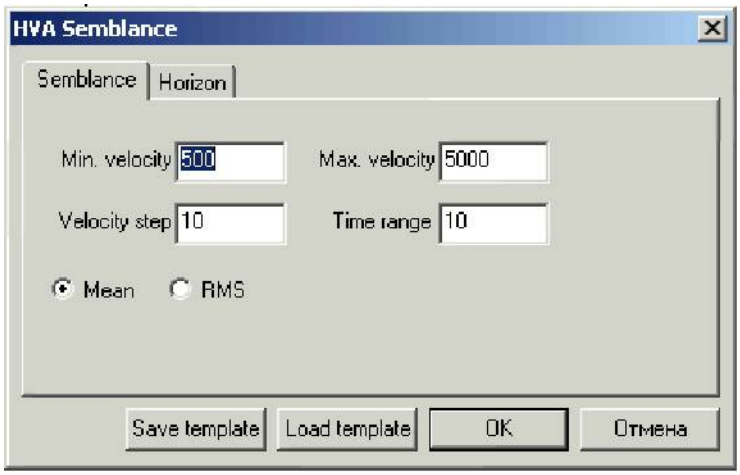
HVA Semblance (Horizon Velocity Analysis Semblance)

This module is meant for velocity field calculation from the specified horizon.

The traces input to the model are to be sorted by CDP and without normal move-out

correction.

When this module is activated the window containing two tabs: **Semblance** and **Horizon** appears.



On the **Semblance** tab you can set the following parameters:

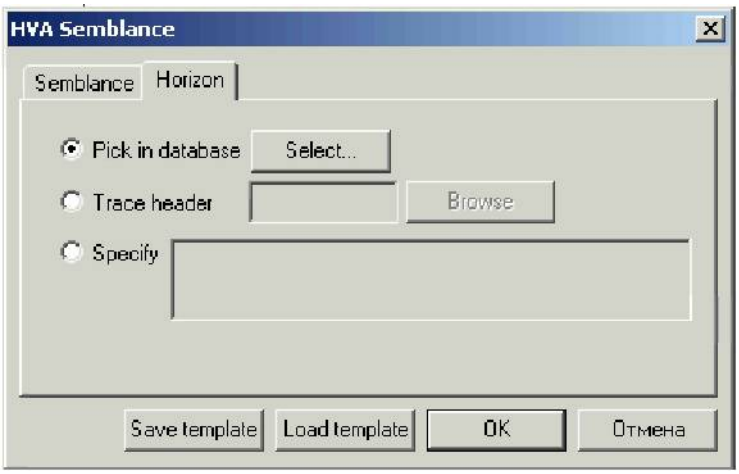
In the **Min. velocity** field the minimum velocity is to be specified. In the **Max. velocity** the maximum velocity is to be specified. Velocity sampling step to be specified in the **Velocity step**

window, the time sampling step to be specified in the **Time Range** window.

The **Mean** and **RMS** options are meant for semblance calculation:

- **Mean** - mean function value in the point to be calculated;
- RMS - root-mean-square function value in the point to be calculated.

On the **Horizon** tab, specify horizon:



There are three ways to do this:

- **Pick in database** - specify horizon from the database. To do this, activate the option, click the **Select...** button and, in the opened dialog box, select the required project database object.

- **Trace header** - specify horizon from the header in which it has been saved before. To do this, activated this option and click the **Browse...** button, then select the required header in the opened dialog box.

- **Specify** - specify horizon manually in the **Specify** field.

The order is the following:

Example:

CDP

100:2250

In this example:

100 - CDP number,

2250 - time.

Save template and **Load template** buttons are meant for current module parameters saving in the template in the project database and for saved parameters loading from previously saved template, respectively.

Usage (An example of module usage in the flow)

Super Gather

Apply Statics

Bandpass Filtering

Amplitude Correction

HVA Semblance

Trace Output

Screen Display

The **Super Gather** module allows creation of the trace sets (super gathers) composed of several CDPs. It is used for signal/noise ratio increasing thus enabling more accurate velocity field calculation.

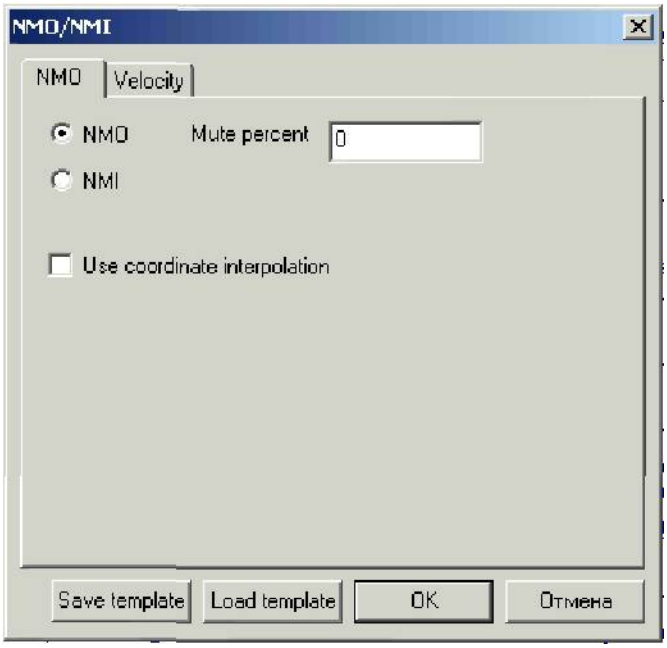
- **Apply Statics** - applies static corrections to data,
- **Bandpass Filtering** - applies bandpass filtering to data,
- **Amplitude Correction** - applies amplitude correction to data,
- **HVA Semblance** - horizon velocity analysis semblance module itself,
- **Trace Output** - saves calculated horizontal spectrum into database of the program,

- **Screen Display –** visualized the spectrum on the screen.

NMO/NMI

This module allows to calculate and to apply normal moveout corrections to CDP trace samples by means of linear interpolation with the help of velocity function.

When this module is activated the following window containing two tabs: **Velocity** and **NMO** appears:

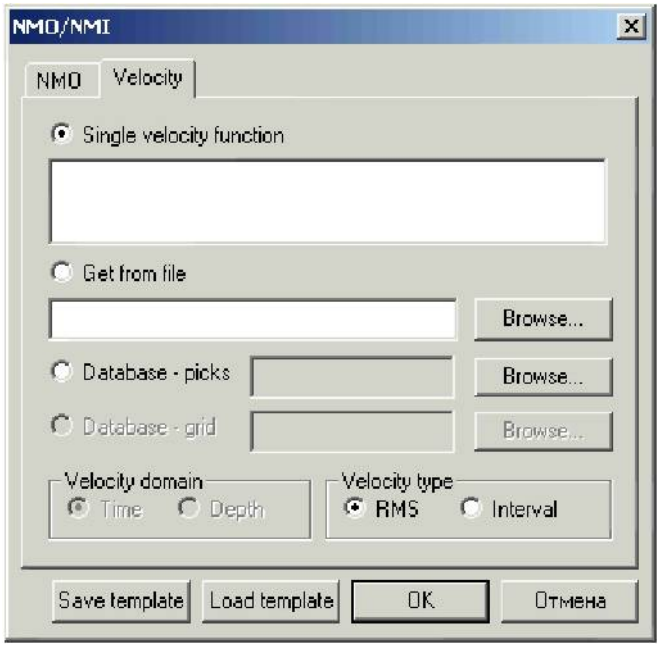


On the **NMO** tab, specify the normal moveout corrections calculation method:

- **NMO** - select this option if it is requited to apply normal moveout corrections to trace samples.

- **NMI** - select this option if it is requited to accomplish velocity inversion, i.e. to apply inverse kinematic function to the seismograms to which the normal moveout corrections have already been applied.

- **Mute percent** - muting parameter expressed in percents. The trace stretching after NMO application is an unwanted but unavoidable result. It is required to set the muting parameter in percents in order to mute all data that has stretched for more then indicated number of percents.



On the **Velocity** tab velocity function specification parameter are grouped:

There are three ways to specify velocity function:

- Activate the **Single velocity function** (constant velocity function) option and specify it manually. The order of specification is the following:

time:velocity, time-time:velocity etc.

Velocities here are in km/s.

- Activate the **Get from file** option and specify velocity function from file. To do this, click the Browse... button, select the desired file in the opened standard dialog box.

- Activate the **Database - picks** option and specify velocity function previously saved in the project database. To do this, click the Browse... button, select the desired database object in the opened standard dialog box.

In the **Velocity type** field it is required to define the type of velocity:

- RMS - root-mean-square velocity,

- **Interval** - interval velocity.

Save template and **Load template** buttons are meant for current module parameters saving in the template in the project database and for saved parameters loading from previously saved template, respectively.

HVA (Horizontal Velocity Analysis)

The module is designed for estimation/improvements of velocity profile along the given horizons.

Principle of operation

The user picks one or several horizons on a stack. Seismic gathers averaged on the given base are formed from CDP gathers along each horizon in the given time window on the basis of the starting velocity curve. Velocity spectrum is calculated from those seismic gathers in the indicated range of velocity parameter variation and is visualized graphically. Further on the user can pick velocity value along the selected horizon on the basis of velocity spectra, analyzing the changes in the appearance of the stack, calculated on the basis of the current velocity function and changes in the appearance of seismic gathers, corrected for normal moveout as well on the basis of the current velocity function. The acquired velocity function can be saved to a database. Other horizons are processed in a similar way.

Input data

The input data for the module represent CDP gathers, CDP stack, starting velocity function (represented in the form of vertical velocity picks or a uniform function) and a set of parameters. All data are input from the database; nothing could be input from the flow or conveyed to the flow.

Output data

Data, which were generated as a result of module operation or adjusted horizontal velocity functions along horizons can be saved to a database by the user during the module operation using the corresponding menu items. A warning message appears when the program operation is finished and you have any unsaved horizontal velocity functions.

Module parameters

The parameters dialogue of the module includes several tabs. Figure (Fig 1) represents the tab of main execution parameters.

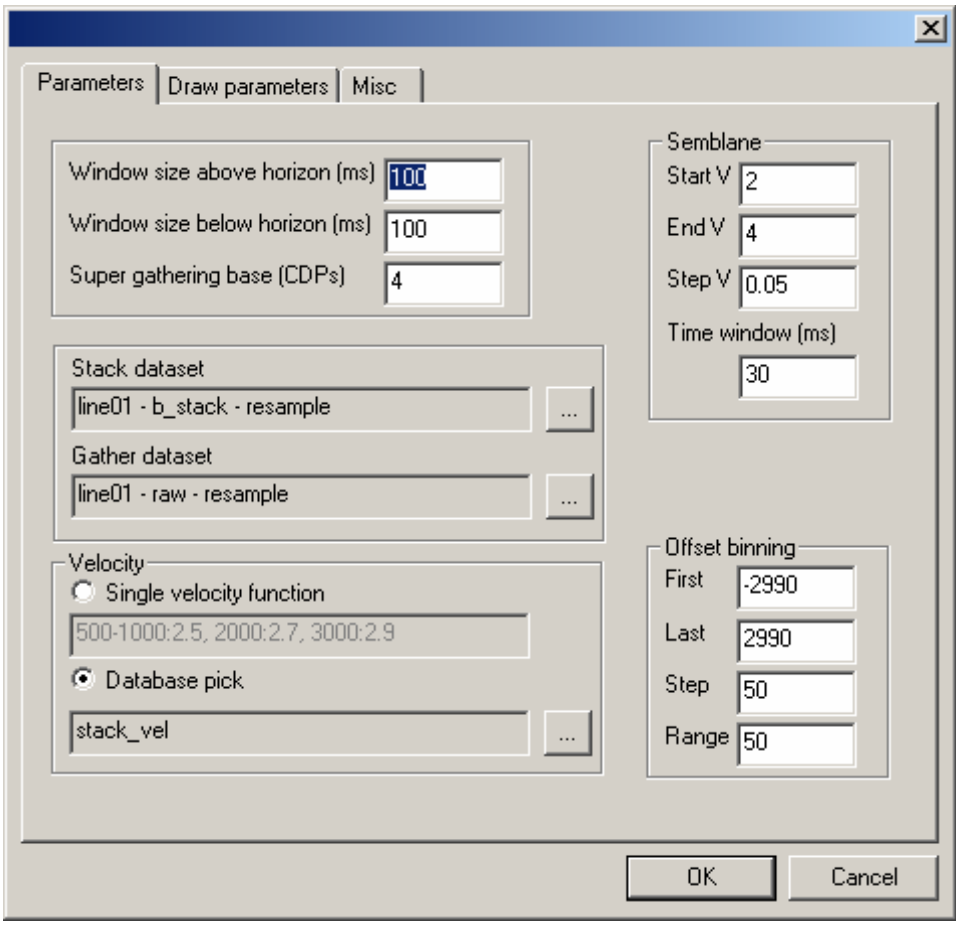


Fig. 1. Dialogue of module parameters, Parameters tab

- **CDP gathers for one profile (Gather dataset)**

Specified by a database object, that contains seismic data (dataset). All CDP seismic gathers that belong to the profile. A CDP header value is considered to be a point number. Before using the data they must be sorted by CDP. If there is a blank in numbers, due to the lack of data, the program will populate the blanks with empty traces.

- **CDP stack for the same profile (Stack dataset)**

Specified by a database object. A dataset should contain the same number of CDPs as the dataset for which seismic gathers are specified. If there is a misfit a user received a warning, and the module

aborts the operation in off-normal mode.

- **Starting velocity function (Velocity)**

Starting velocity function should be specified using one of the following modes:

- **Single Velocity function**

A line written using a common **RadExPro** format.

Example: 500-1000:2.5, 2000:2.7, 3000:2.9

- Vertical wave field, specified as a database object (***Database pick***)

- **Other parameters**

- Super gathering base (***Super gathering base***, in number of gathers)

- Sizes of time windows below and above the horizon, within which the super gathers are generated, and the stack is calculated are specified by two parameters - a part of window, situated above the horizon (***Window size above horizon***) and a part of window below the horizon (***Window size below horizon***).

- Parameters of velocity spectrum calculation (group of parameters ***Semblance,*** parameters ***Start V*** and ***End V*** – start and end velocity for search (in km/s), step of search ***Step V*** and time window size ***Time window***, the samples of which are used in velocity spectrum generation)

- Offset binning parameters (***Offset binning***) include minimum and maximum offset (***First, Last***), offset step (***Step***) and range of grouping window (***Range***), all parameters are in meters.

Draw parameters tab (Fig. 2) includes parameter groups, that are related to visualization mode of data, velocity function and semblance.

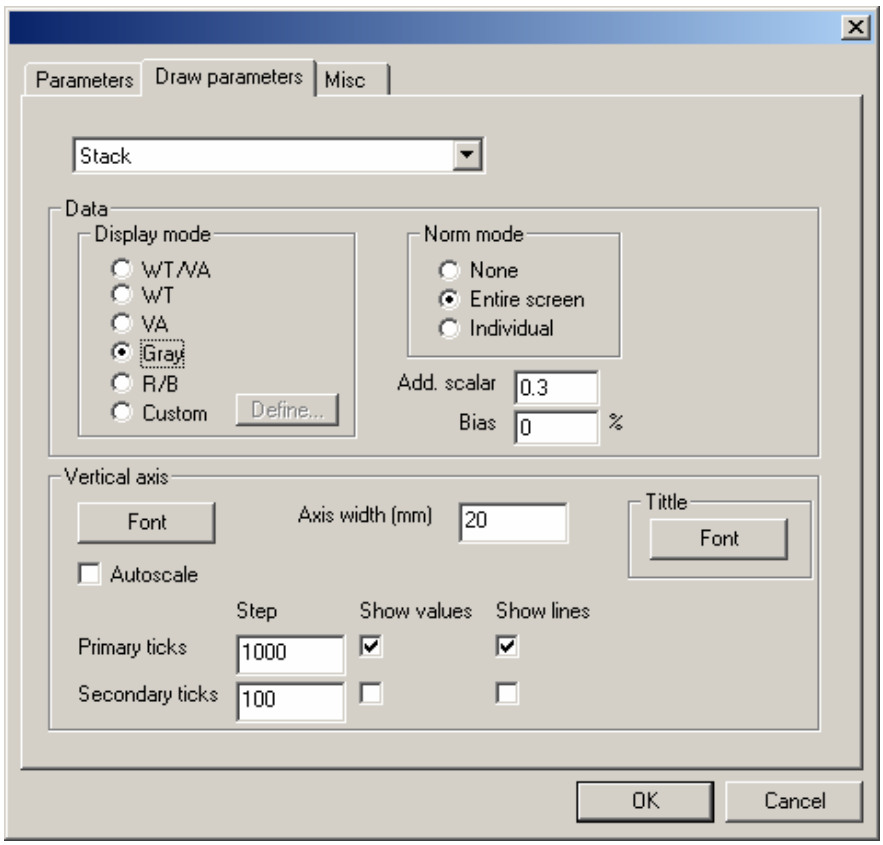


Fig. 2. Module parameters dialogue, Draw parameters tab

Using the tab you can specify visualization parameters of different data components, used in velocity analysis:

- Stack (**Stack**)
- Initial velocity function (**Model**)
- Initial stack strip along the horizon selected by the user (**Initial stack strip**)-
- Stack strip along the horizon selected by the user, acquired during the module operation with the current velocity function (**Modif. stack strip**)
- Velocity spectrum (**Semblance**)
- Super gathers (**Super gathers**)

- Horizontal axes, one for all elements (**Horizontal axis**).

You can specify **Display mode** and **Norm mode**, screen gain (**Additional scalar**) and **Bias**, the parameters' context is the same as in **Screen Display**.

- **Additional scalar**. (screen gain) - Additional scalar for multiplying samples' values before display.

- **Bias**. Bias of an average trace level from zero, that leads to change in black color level in case of the variable area display. Positive value will cause a leftward shift of the trace zero line and an increase of the blacken area under the curve. A negative value corresponds to a decrease of the blacken area under the curve

- **Display mode**. Select the mode of traces' display. Possible variants:

- **WT/VA** displays traces using the wiggle trace mode/variable area mode,

- **WT** displays traces using the wiggle trace mode,

- **VA** displays traces using the variable area mode,

- **Gray** displays traces using the variable density in grey palette mode,

- **R/B** displays traces using the variable density in red-white-blue palette mode,

- **Custom** displays traces using the variable density in palette, specified by the user,

Define button is available if the option **Custom** is on. It opens the dialogue window **Custom Palette**.

Vertical axis display parameters can be specified in group **Vertical axis** for each element of velocity analysis window individually. (Horizontal axis' parameters are set simultaneously for all elements

-- select **Horizontal axis** in the drop-down list at the top of the dialogue page.)

For an axis you can change font of the tick values (**Font** button), font of the axis title (**Title: Font**), **Axis width**, the intervals (**Step**) for **Primary ticks** and **Secondary ticks**, and the option of whether to show values (**Show values**) and corresponding lines (**Show lines**) or not.

The last tab (**Misc**) of the parameters dialogue offers selecting the **Supergather display step** during velocity analysis and indicating whether disk buffering is needed for saving intermediate results when calculating velocity spectra (**Use disk buffering while processing**).

The lattermost mode is worthwhile if and only if the data volume is big and the module can't function without it. Otherwise, using this option will simply slow-down the operations.

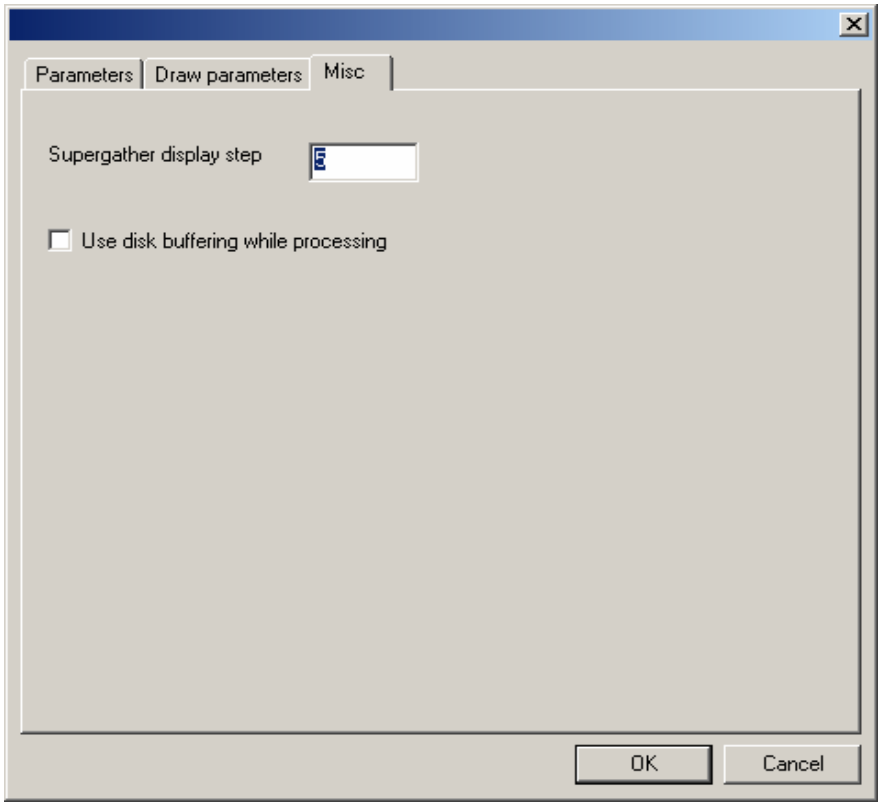


Fig. 3. Parameters dialogue of the module, Misc tab

User interface and functionality

- **Parameters' setting and module activation**

The module is standalone, i.e. it doesn't receive seismic data from the flow, and it operates through

a database. At the same time the modes of representation and running don't differ from ordinary modules. Parameters are specified through parameters' dialogue window, the module is run using the button **Run** in the main application menu.

All necessary parameters should be specified in correct way; otherwise the module will not start operation, displaying a message on a corresponding error.

- **Program configuration**

The program by default consists of two working windows.

- **Stack display window and initial velocity model** (Fig. 4).

This window in turn has two sub-windows:

- stack display window

Input stack is displayed in this window. A user can perform horizon picking and select a horizon for velocity analysis.

- Initial velocity function display window

Initial velocity function is displayed in this window in time domain.

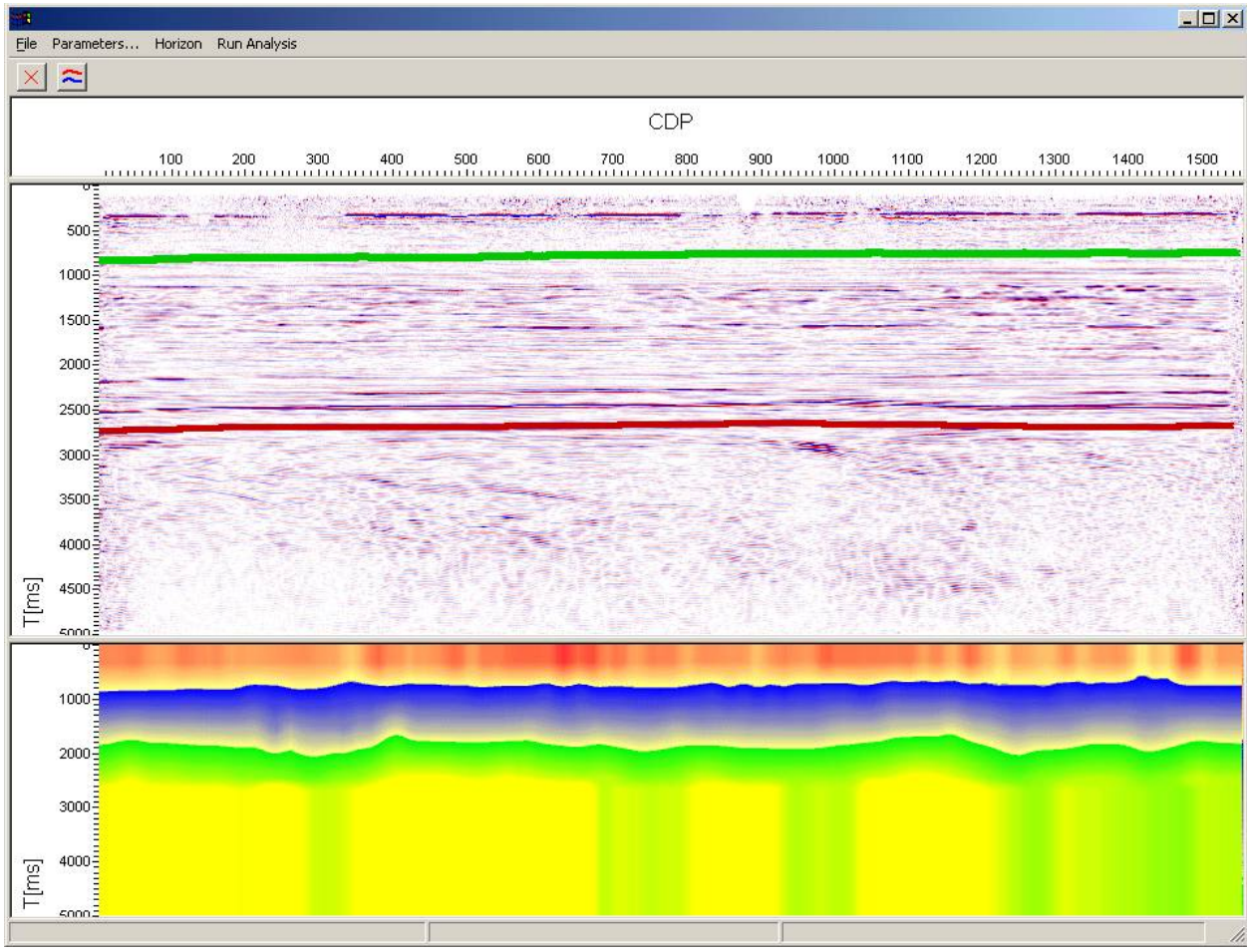


Fig. 4. Stack display (upper part) and initial velocity model (lower part) windows

- **Window for velocity analysis along the selected horizon.**

This window is opened when running velocity analysis on selected horizon. It consists of four parts (Fig. 5, from top to bottom).

- Initial stack display window along the horizon.
- Modified stack display window along the horizon.
- Velocity spectra display window
- Supergather set display window.

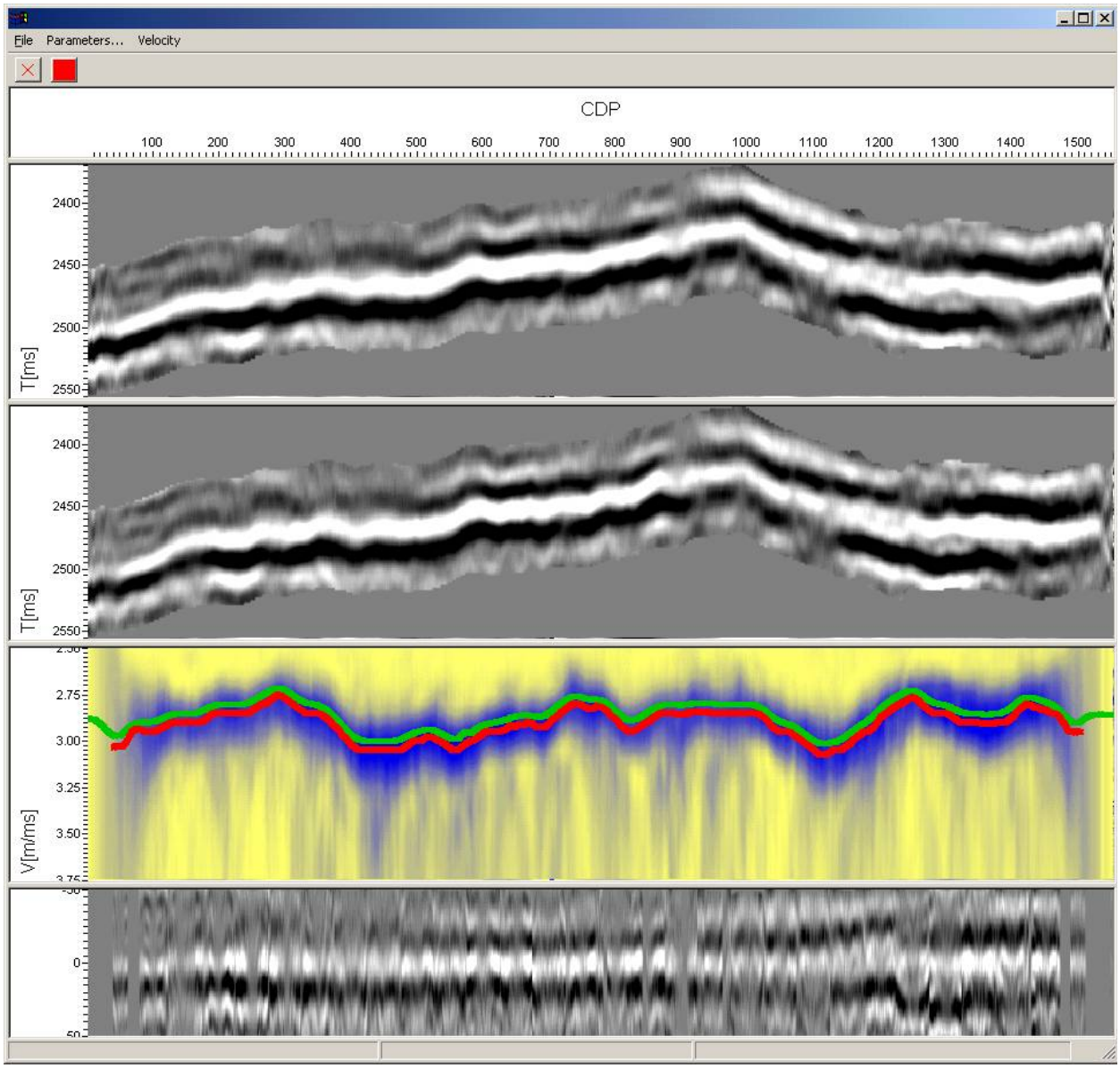


Fig. 5. Velocity analysis window along the selected horizon

All windows of velocity analysis along the horizon support changing the display scale; there are scroll bars when required and display scrolling is allowed using scroll bars. All windows have joint horizontal scale and joint horizontal scroll bar. The windows are synchronized between each other and with the scale and scroll bar in horizontal direction. The windows initial and modified stacks are synchronized in vertical direction as well.

Program control is carried out through menu elements, some of which are duplicated by toolbar buttons. Some parameters can be changed after having run the program.

- **Module running and opening the main window of the module**

When you run the module first of all the main window of module is opened, which consists of two parts. The upper part displays the stack; the lower part displays the initial velocity model. In the upper part there is a horizontal scale on CDP, in each part of window there is a vertical time scale at the left side. If required there are scroll bars, while the horizontal scroll bar, as well as horizontal scale is shared, one for each half-window. Sub-windows and all three scales are synchronized, in the sense that the scale change and scrolling through the screen is performed synchronically. You can perform zoom in/zoom out and modify stack display and initial velocity model parameters.

You can generate/modify/save/load picks, modify velocity analysis elements' display parameters in the main window; as well velocity analysis window can be run from the main window.

The following menu items are reserved for performing these operations:

File/Exit – exit the module

Parameters – displays visualization parameters, is completely identical with the tab **Draw parameters** (see above) of the module parameters dialogue.

Horizon – working with picks.

Picking – switch on/off the mode of horizon tracking (hot button "M")

Mode – mode of horizon tracking (hot button "A") – activates parameters' setting dialogue of manual/automatic tracking.

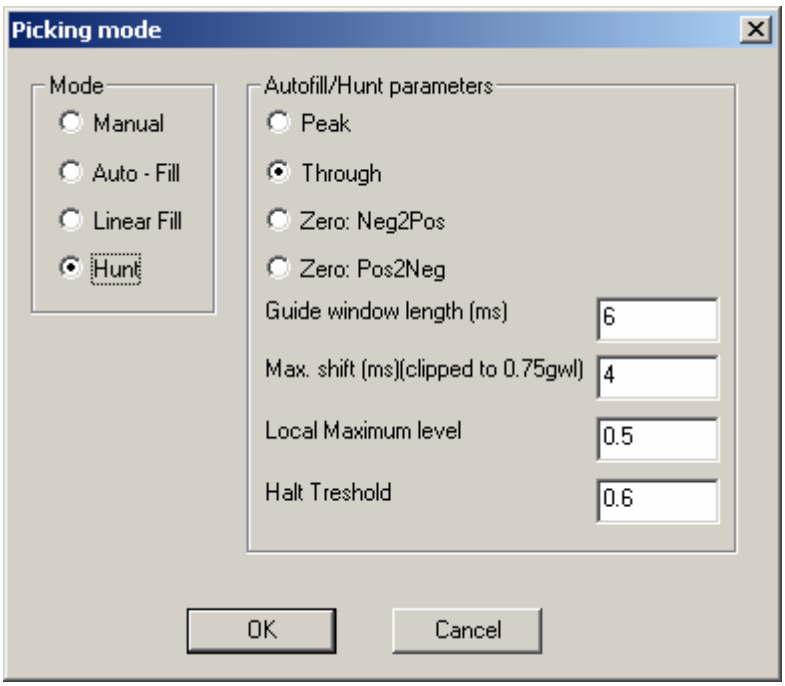


Fig. 6. Parameters' dialogue of the Module

The following operations are available for horizon being edited:

Hunt – automatic tracking of seismic events on the basis of correlation between neighboring traces. Guide window length – window length, in which the cross correlation function between the neighboring traces will be calculated; Max shift – is the maximum allowed shift of the event on the neighboring traces; Local maximum level – the local maximum level with reference to the global extremum; Halt Threshold – threshold of the global maximum of cross correlation function, the tracking is terminated below this threshold.

Auto fill– tracking of phase maximum in the window between two specified points.

Manual – linear interpolation between the indicated points sequence.

New - create new pick (N hot key).

Delete – delete active pick (Delete hot key)

Save as – save current pick to a database, with the name indication (Ctrl+S hot key).

Load – load pick from the database (Ins hot key)

Line style – specify the line style in the dialogue (S hot key)

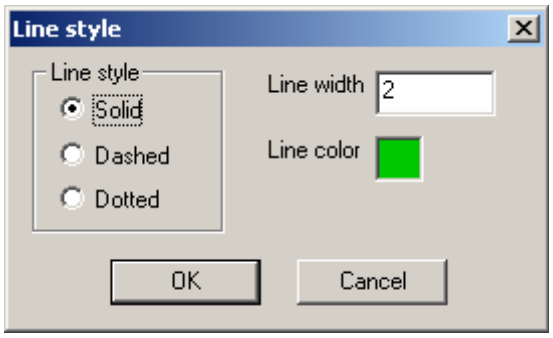


Fig. 7. Line style specification dialogue

Smooth – smooth (averaging in selected window, R).

Next/Previous – toggling between active picks (Tab/Shift+Tab)

After having chosen any horizon you can run a procedure of horizontal velocity analysis using the menu item **Run Analysis**.

You are not allowed to select horizon, on which the velocity analysis is currently being performed, for editing. In order to edit a horizon, on which velocity analysis is performed, you have to exit the mode of velocity analysis and select another horizon for velocity analysis. After that, you may edit the horizon. However, in general, you have to reshape averaged seismic gathers for subsequent velocity analysis along this horizon, as well as re-calculate velocity spectra. After the user had finished editing the horizon and exited the picking mode, it is possible select the horizon for velocity analysis.

- **Operation in the window of velocity analysis on selected horizon**

Velocity spectra display and velocity function picking

After a successful velocity spectrum calculation for selected horizon it is displayed in special graphic window. Vertical axis represents velocity axis, there is vertical scale and vertical scroll bar. Operations zoom in, zoom out, scrolling are available.

This window displays current velocity function above semblance, as well as initial function, used for semblance calculation.

The user can pick velocity function in the semblance display window.

The following operations are available:

- Hunt – tracking the maximum phase in selected time window.
- Fill– tracking the maximum phase in the window between two specified samples.
- Manual – linear interpolation between the specified samples' sequence.
- Smooth – smoothing (averaging in the specified window).

Super gather set display

The fourth window of velocity analysis on selected horizon displays NMO-corrected super gather set.

Each N-th super gather (N – is specified by the user in Parameters) is displayed. The traces are visualized with an equal interval between each other and occupy all space available. Available display parameters: selection of traces visualization mode (WT, VA, WTVA, color with palette) and gain.

Super gathers' position is linked with their central CDP points and synchronized with the stacks and semblance display window.

NMO correction is calculated on parallel traveltime curves on the basis of current velocity for the given CDP. Editing velocity function, causes changed in velocity and subsequent NMO recalculation and redrawing of the corresponding super gathers being displayed.

Calculation and display of modified stack strip. Initial stack strip display

Stacking on parallel traveltime curves is carried out on the basis of current velocity function for each initial gather. It is performed within the time window along the horizon. Thus, a strip of modified stack is acquired along the horizon. This strip is displayed in a separate window. If the user edits velocity function, the stack traces of the CDPs being modified are stacked again using new velocity values.

Moreover a strip is displayed in the same time window along the horizon of the initial stack.

Specific requirements and comments

Mandatory header fields: CDP, OFFSET, DT, NUMSMP.

Fields TLIVE_S and TFULL_S are responsible for muting and contain time values of start and end of "meaningful" samples in ms. Make sure, that they don't contain trash.

Velocity values in Parameters dialogue are given in m/ms.

Flag "Use secondary storage while processing" specifies whether the working data set will be loaded into a file during the analysis of the selected horizon, or the data will be loaded when recalculating characteristics and dropped off immediately. In the second case (flag is specified) the processing /recalculation time is greater, but you need less frame memory.

It makes sense to save the acquired horizontal velocity function to the database after velocity analysis along the horizon (object HVT). There is a utility function **Hvt→Vvt** that helps to make a transformation of a set of horizontal velocity functions into a vertical velocity field (menu **Tools**). As well there is a possibility to export horizontal velocity function from the database to a text file, via the Database Manager.

Transformation of horizontal velocity functions' into vertical using HVT→VVT utility

Access to utility of transformation horizontal velocity functions into vertical functions is carried out through the menu **Tools** of the main window **RadExPro**.

The appearance of utility function window is displayed on Figure (Fig. 8).

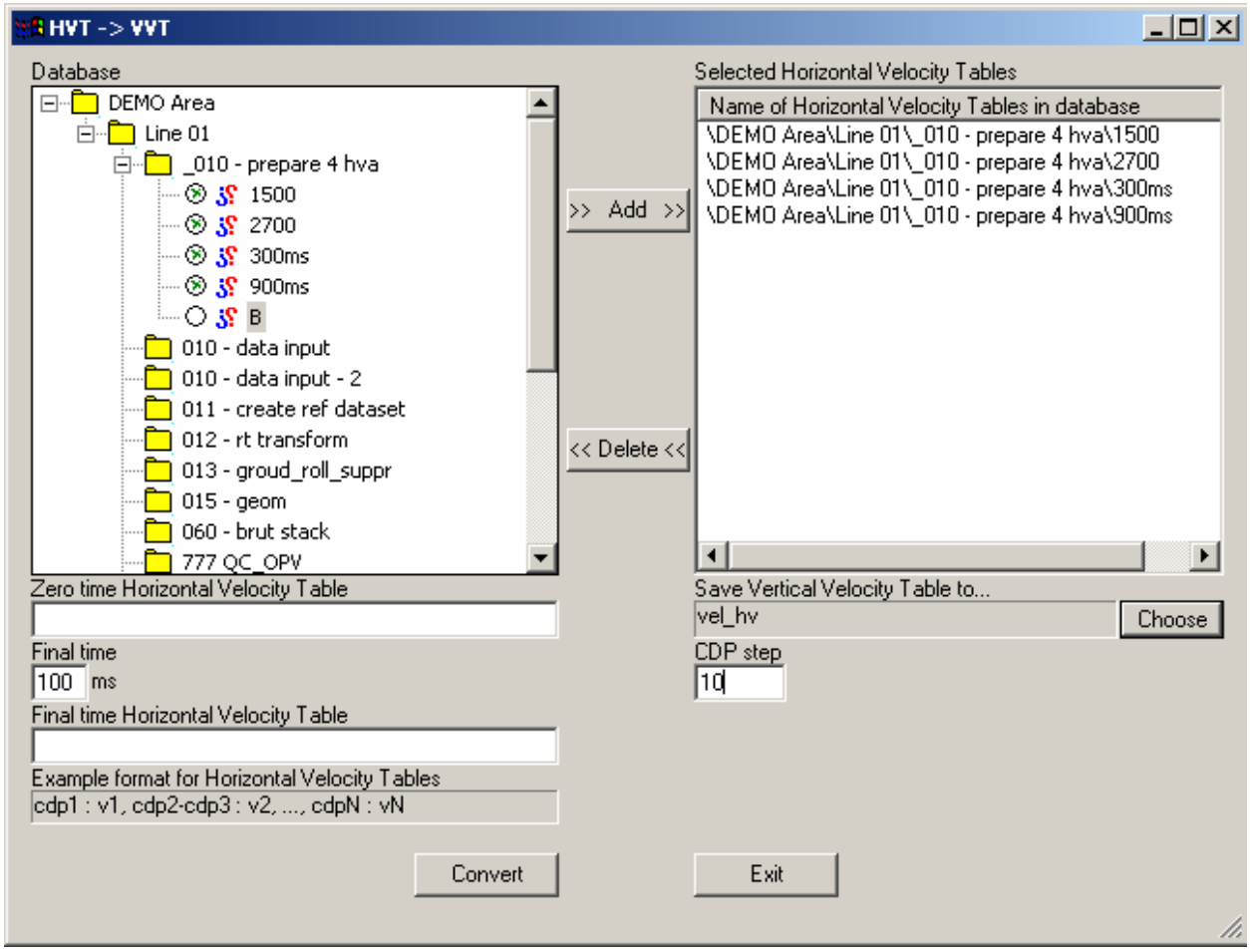


Fig. 8. Utility of transformation horizontal velocity functions into vertical HVT→VVT

To convert a set of horizontal velocity picks (acquired as a result of **Horizontal Velocity Analysis**) into vertical, a user should perform the following:

- Run the utility.

- Select one or several horizontal picks (saved in the database and displayed on the left panel **Database**) and add them to the set, displayed at the right panel **Selected Horizontal Velocity Tables** using the buttons **Add** and **Delete** (the button removes the selected pick from the set, it doesn't delete it from the database).

- Optionally the user can specify velocity value at 0 time (**Zero time Horizontal Velocity Table**) and at time value (**Final time**), specified by the user (**Final time Horizontal Velocity Table**)

- Specify CDP step for the vertical velocity picks (**CDP step**)

- Select an object from the database (vertical velocity pick), to which the pick will be saved, clicking the button **Save Vertical Velocity to… Choose**
- click the button **Convert**.

Velocity Editor

This module is meant for creation of two-dimensional velocity models, as well as their editing, smoothing, and binding to coordinates.

The module interface consists of two parts: dialog box for initial parameters setting, and a primary running window of the module during its execution. The dialog window for parameters setting can be opened during module operation, but in this case all parameter changes will be active only during the current module execution session.

Velocity editor is applied on manual manipulations with the velocity field. In the case of single-channel investigations, when the velocity field cannot be obtained directly from the data, the interactive velocity editor is the only way to specify the velocity field manually using external information.

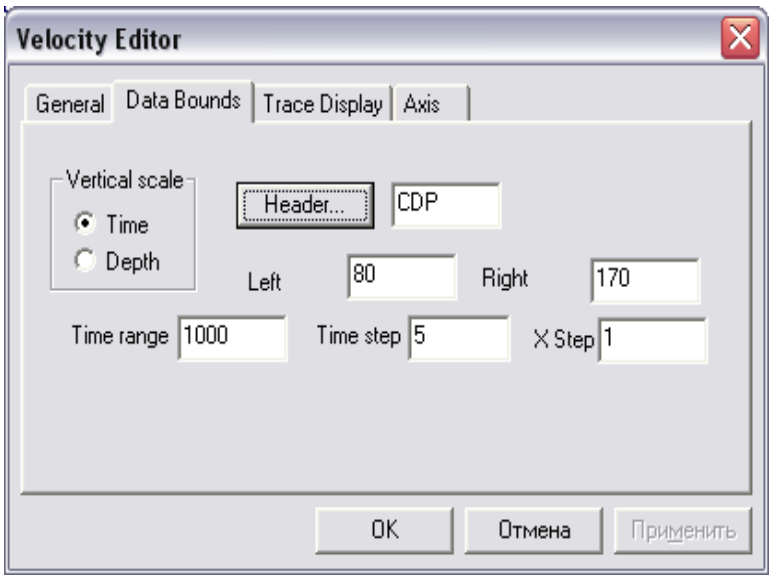
The velocity field is constructed by means of interactively defined polygons filled by certain velocity values and with the help of a brush used to for color the velocity field manually.

In the flow, a velocity editor can operate on its own as an independent application. In this case in the module window only the velocity field is displayed and being edited.

If the *Velocity Editor* is launched from the flow in which the stacked data are running, the velocity field will be displayed against the time section background in the module window. The velocity field is matched to the data according to a coordinate (distance) along the line, which shall be stored in the data in an arbitrary trace header field and correspond to the coordinates specified in the velocity field file.

Parameters

The dialog window for the initial parameter specification for the module is as follows:



The dialog window contains four tabs.

The **General** tab contains parameters which define the position of files with polygons and velocity field on disk.

- **Velocity file:** Specifies the name for the text file that contains (or will contain) a velocity field. To select a file, click the **Browse...** button or enter the file name manually.

- **Polygons file:** Specifies the name for the text file that contains (or will contain) the polygons description. To select a file, click the **Browse...** button or enter the file name manually.

The **Data bounds** tab contains parameters that define velocity field size.

In case, there is an input dataset passing through the flow, click the **Field** button to define the trace header field of the input dataset that contains the linear coordinate for matching with the velocity field. In case, there is no input dataset available, this field will be ignored and all coordinate values will be in abstract units.

- **Left bound:** coordinate of the left velocity field edge

- **Right bound:** coordinate of the right velocity field edge

- **Time range:** Time range (maximum time) of velocity field

- **Time step:** Vertical step of velocity field grid

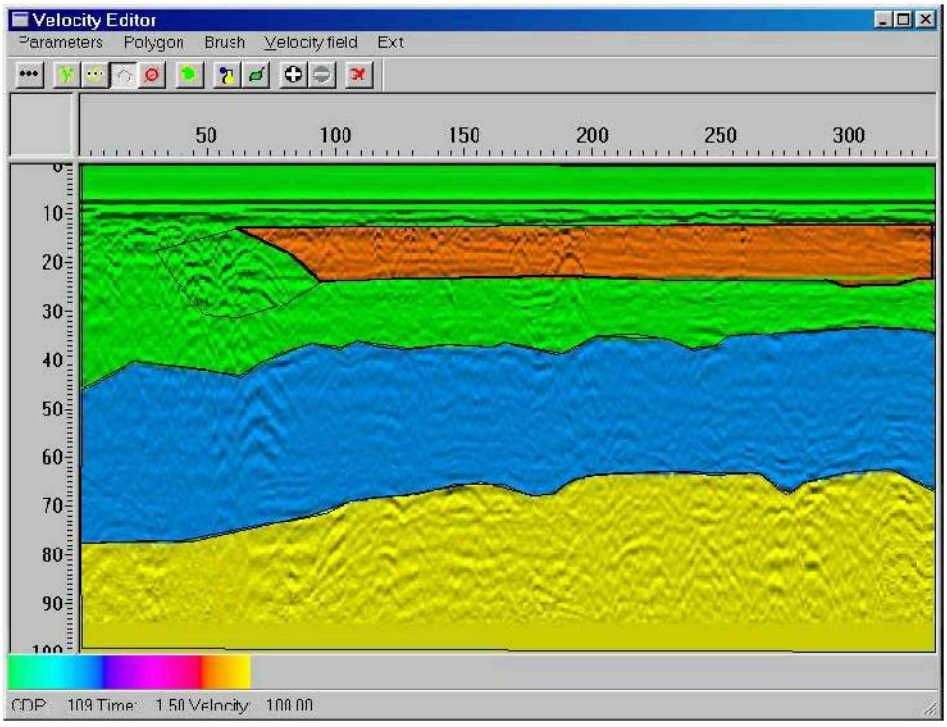
- **X step:** Horizontal step of velocity field grid

The **Trace display** tab contains parameters that affect the data visualization method. They make sense only if some stacked data are running in the flow in which the module operates. The meaning of these parameters is the same as that of parameters from *Screen Display* (the **Common Parameters** dialog window) module dialog box.

The Axis tab contains two fields (X Step and Time Step) where the intervals between labels on the coordinate axes should be specified.

Working with the module

The running window of the module is similar to the following:



The working area of the **Velocity Editor** module consists of:

- The menu in the upper part of working area
- The toolbar below the menu
- Horizontal and vertical coordinate axes
- Data fields

- Palettes
- Status bar at the bottom of the working area
- Scrollbars which appear in case if the image is enlarged

Menu and toolbar

The toolbar and the menu contain the following commands:

- • **Parameters/Parameters...** Allows initial module parameters changing, (see **Parameters** chapter).

- • **Polygon/New** Creates a new polygon. For new polygon the editing mode is automatically switched on.

- • **Polygon/Edit** Switches on/of the edit mode for active polygon. In case if the edit mode is switched off, the polygon can be made active by clicking the left mouse button (**MB1**) on one of its points. The polygon edit mode makes it possible to add/delete/relocate polygon points, close and open the polygon. **To add a point to a polygon,** click the left mouse button (**MB1**) at the desired spot. The points are sequentially added to the polygon.

To relocate the point, capture it by clicking the right mouse button (**MB2**) and drag it to the desired place. When clicking and holding the right mouse button (**MB2**) the active polygon point closest to the cursor position will be captured.

To delete the point, click the left mouse button (**MB1**) near the point while simultaneously holding the Shift key down. The active polygon point closest to the cursor position will be deleted.

- **Polygon/Close** Closes/Opens the polygon. This command is available only within polygon edit mode.

- **Polygon/Delete** Deletes active polygon.

- **Polygon/Fill...** Fills the active polygon with specified velocity values, possibly with horizontal

and/or vertical velocity gradient. To fill in the polygon, close it first. When this command is chosen the dialog box for specification of velocity and two gradients appears.

- **Brush/Parameters** Serves to set the "velocity brush" parameters. When this command is chosen the dialog box for "velocity brush" parameters specification appears. Here, specify the shape, size and velocity of the brush.

- **Brush/Brush** Serves to switch the mode of model drawing by "velocity brush" on or off. In the brush mode the oval or the rectangular (according to selected brush shape) are displayed at the cursor position point. With the left mouse button (**MB1**) held down, when moving the mouse over the area covered by the brush, the velocity specified in brush parameters will be set.

- **"+"** Serves to switch on/off the image zoom-in mode. In the zoom-in mode the mouse cursor changes its shape. By clicking the left mouse button (**MB1**) you can enlarge the image by one step. When doing this, the image is scrolled so that the selected point is in the center of the screen.

- **"-"** Serves to switch on/off the image zoom-out mode. By clicking the left mouse button (**MB1**) you can diminish the image by one step.

- **Exit** Exits the velocity editor. When setting parameters, if the files names for the velocity and polygon fields were specified, then the changes will be automatically saved when exiting.

Interactive Velocity Analysis

The **Interactive Velocity Analysis** module is used for interactive analysis of the stacking velocities.

The following header fields should be filled in correctly:

- *SCDP* – CDP supergather number. If the data input into the flow is performed using the module **Super Gather**, this field is assigned automatically. Otherwise it should be assigned manually.

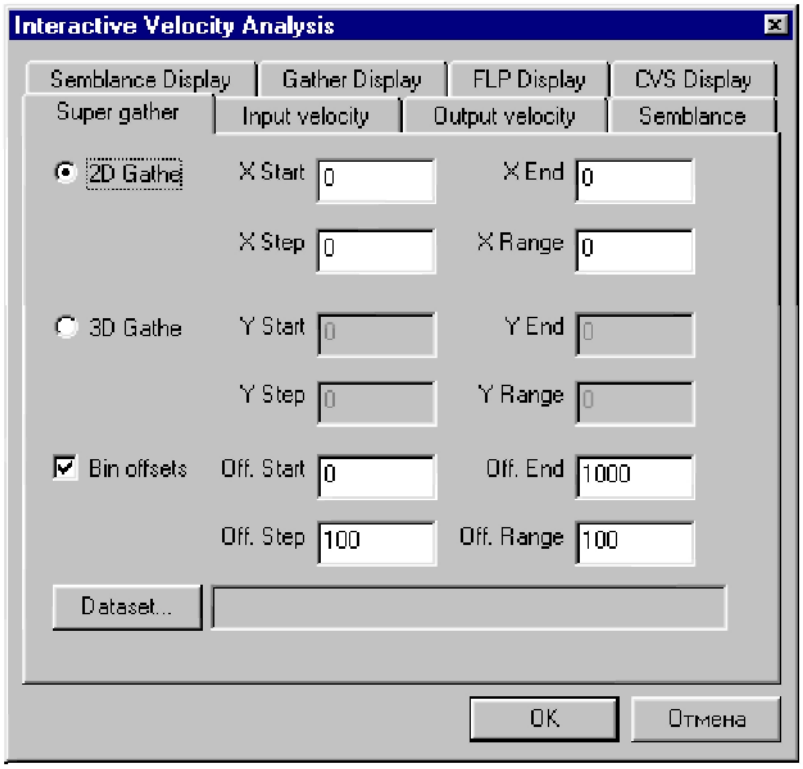
- *OFFSET* and *AOFFSET* – should contain offsets and their absolute values, correspondingly.

- *ILINE_NO* and *XLINE_X* – for 2D data, the first field should contain the CDP numbers (*CDP*),

the second field – any similar integer number.

The module interface consists of two parts: a window for initial parameters setting and a main window of the module that appears while executing this module. The dialog box for parameters setting can be opened while the module is running but, in this case, all parameter changes will be active only during the current module execution session.

When the module is activated, the window containing 8 tabs will open:

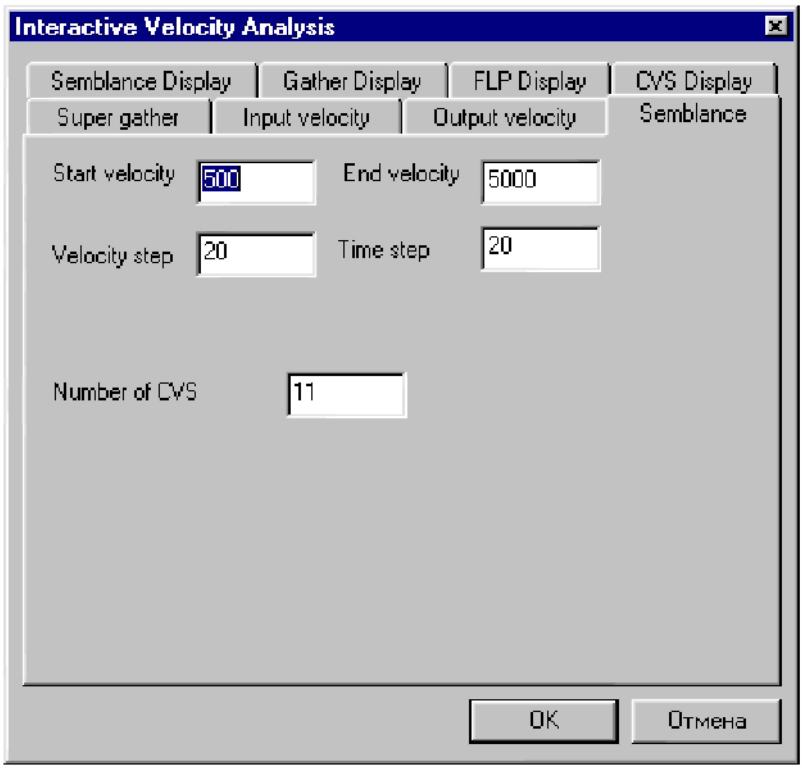


- **Output velocity.** Here, specify a database velocity pick object (recommended) or a text file that will contain the resulting edited velocities. This tab is similar to the window that appears when the *NMO/NMI* module is activated, except for the **Single velocity function** option which is lacking in the tab.

- **Input velocity.** Here, specify an existing input velocity field (a velocity pick object in the database or a text file) to continue editing it. In case, the module is executed for the first time and no velocity function exists yet, specify the same velocity pick object as in the **Output velocity** tab. As a result, after you execute the flow with the module for the first time and save the velocities, for the following sessions the same velocity field will be used as an input. Thus, you will be able to continue editing it. This tab is similar to the window that appears when the *NMO/NMI* module is activated.

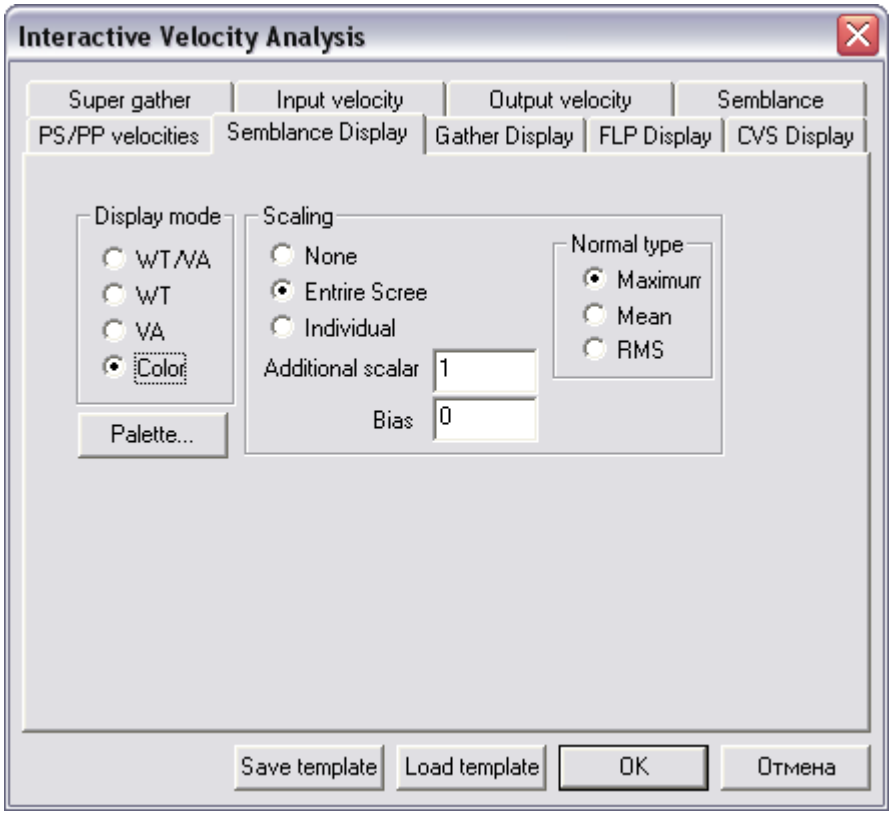
- **Super Gather.** On this tab, only the **Bin offsets** similar to that from *Super Gather* module parameters option is available.

- **Semblance.** This tab sets the parameters for semblance (velocity spectrum) calculation as in the following:



In the **Start velocity** and **End velocity** fields the start and end velocities for velocity searching should be specified. To calculate the semblance the velocity search step should be set in the **Velocity step** field, and the time search step in the **Time Step** field. The number of constant velocity stacks displayed on the screen is specified in the **Number of CVS** field**.**

- **Semblance Display.** This tab is used for setting the velocity spectrum (semblance) display parameters as in the following:



In the **Display Mode** field select the display mode for the traces of velocity spectrum:

- WT/VA displays the traces in wiggle trace and variable area mode
- WT displays the traces in wiggle trace mode,
- VA displays the traces in variable area mode
- **Color** displays the traces in variable density modeBy clicking the **Palette...** button you can specify the color palette manually (when all color palette point are deleted, the traces are displayed in grayscale).

In the **Scaling** field, set trace amplitudes normalization

- **None** - there is no additional trace normalization
- **Entire screen** - all traces are normalized in the aggregate by means of dividing the trace

amplitudes by mean absolute value of amplitude of all traces

- **Individual** - normalizes all traces individually by means of dividing the trace amplitudes by mean absolute value of amplitude of the trace itself

- **Normal type.** In this the type of amplitude normalization to be set

- **Maximum** - according to maximum value

- **Mean** - according to mean value

- **RMS** - according to root-mean-square value.

- **Additional scalar.** In this field, enter an additional scalar (gain factor) on which the trace samples values will be multiplied before being displayed on the screen.

- **Bias.** In this field, specify a value of average trace level bias from zero that results in black color level changing when variable area mode is used for visualization. The positive value will cause displacement to the left from the zero line of the trace and enlarge the blackened area of the curve. The negative value corresponds to reducing of the blackened area of the curve. When **Color** display is selected, this value will shift the zero level of the data relative to the center of the color palette.

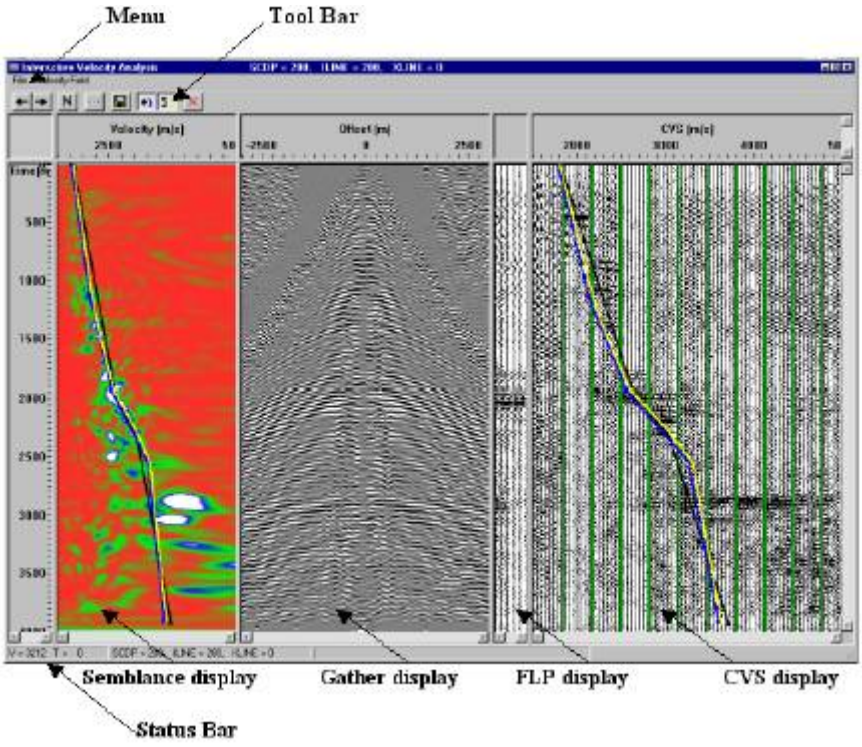
- **Gather Display.** This tab is used to set seismogram display parameters. The tab looks the same as the **Semblance Display** tab.

- **FLP Display** This tab is used for the setting of display parameters of the dynamic stack obtained via the current velocity function picked in the **Semblance Display** panel. The tab looks the same as the **Semblance Display** tab.

- **CVS Display** This tab is used for the setting of display parameters for constant velocity stacks. The tab looks the same as the **Semblance Display** tab.

Working with interactive velocity analysis module

When starting execution of the flow containing the **Interactive Velocity Analysis** module, a window similar to the one shown below will open:



The window is subdivided into 5 parts:

- Vertical scale of wave two-way traveltime (expressed in ms)

- **Velocity** - the panel of velocities spectrum obtained via semblance calculation(the parameters setting for this panel is done on **Semblance display** tab)

- **Offset** - seismogram display panel (the parameters setting for this panel is performed on **Gather display** tab)

- The panel for the dynamic stack obtained with the current velocity function picked on the semblance (the parameters setting for this window is done on the **FLP display** tab)

- **CVS** - constant velocity stack panel (the parameters setting for this panel is performed on **CVS**

display tab).

All the panels have one common Y axis (time axis, ms) but every panel has its own X axis: velocity axis for **Semblance display** and **CVS display,** offset axis for **Gather display.**

Velocity function picking is accomplished in the semblance panel **(Velocity** or **Semblance display).**

Point adding is done by clicking the left mouse button (**MB1**) on the desired point.

Relocation of existing points is done by clicking the right mouse button (**MB2**). By doing this the point closest to the cursor position will be relocated.

Point removal is performed by double-clicking the right mouse button (**MB2**) on the point while holding down the Ctrl key (Ctrl+**MB2** DblClick).

In order to enlarge a data display fragment horizontally or vertically, place the mouse cursor in the desired axis area and click the left mouse button (**MB1**) on the point corresponding to one of the edges of the fragment to be enlarged. Then, holding the button pressed, relocate the cursor to the point corresponding to the other edge of the fragment. After releasing the button the selected data fragment will be enlarged to the size of the whole panel.

In order to return to the initial scale on one of the axes, double-click the left mouse button (**MB1**) in the corresponding axis area.

Menu and Tool Bar

Menu

The menu contains the following submenus:

- **File**

- **Exit** - exits the **Interactive Velocity Analysis** module.

- **Velocity Field**

Show previous: displaying the previous velocity function in the **Semblance Display**

Show mean: allows displaying a mean velocity function in the **Semblance Display**

Show Lay: allows displaying layer velocities calculated from Dix formula in the **Semblance Display**

- **NMO**

Muting Percent allows stretch muting parameters specification in percentage. All data that has stretched as a result of NMO on more than specified percentage will be zeroed.

Tool Bar

At the top of the running window there is a tool bar:

The following commands are available (from left to right):

- switching to the next seismogram

- switching to the previous seismogram

- switching on the interactive NMO correction of the current seismogram in the **Gather Display** in accordance with the velocity function set in the **Semblance Display**panel

- launching of **Interactive Velocity Analysis** parameter dialog where you can change the initial parameters set during flow creation.

- saving the changes into a file or database depending on the settings specified in the **Interactive Velocity Analysis** parameter dialog.

- displaying of the previous velocity function in the **Semblance Display.**
- displaying of the mean velocity function in the **Semblance Display.**
- displaying of layer velocities calculated from Dix formula in the **Semblance Display**
- deleting of all points of the active velocity function pick
- exiting the **Interactive Velocity Analysis** module

Status Bar

At the bottom of the **Interactive Velocity Analysis** window there is a status bar.

In the left field of the status bar you can see information about velocity (V, m/s) and Time (T, ms) at the point of the current cursor position. In the central field you can find information about the current supergather (SCDP) and profile numbers (in 2D case) or about InLine, CrossLine numbers (in 3D case). The information contained in the right field of the status bar is duplicated in the header bar of the module running window.

Example of module application in the flow

A typical flow with interactive velocity analysis may look like the following:

- Super Gather
- Apply Statics
- Bandpass Filtering
- Amplitude Correction

- Interactive Velocity Analysis

- **Super Gather:** creates the trace sets (supergathers) containing several CDPs. It is used to increase the signal/noise ratio and, therefore, for better velocity analysis accuracy

- **Apply Statics:** applies statics to the data

- **Bandpass Filtering:** applies bandpass filtering to the data

- **Amplitude Correction:** applies amplitude correction to the data

- **Interactive Velocity Analysis:** the interactive velocity analysis itself.

IMPORTANT! In **the Super Gather** module the **Bin Offsets** option should be switched off. When needed, binning parameters may be set in the **Interactive Velocity Analysis** module.

Launching the module from Map application

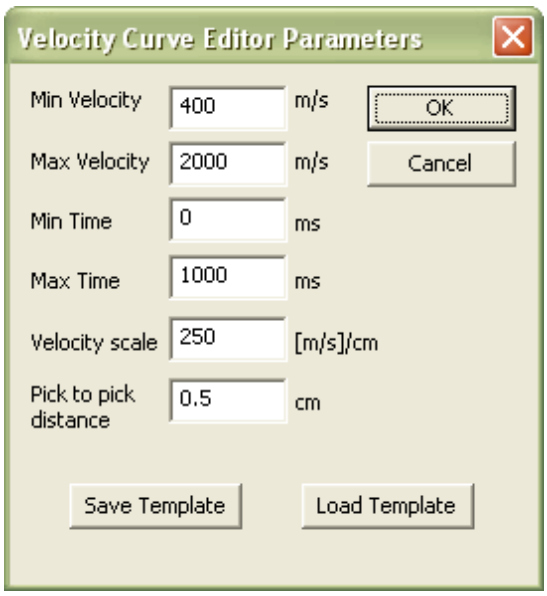
The interactive velocity analysis module **(Interactive Velocity Analysis)** can be launched from

the **Map** application by clicking the **V** button on the tool bar. All the parameters for the module should be specified in the **Interactive Velocity Analysis** parameter dialog.

Velocity Curve Editor

Velocity Curve Editor is designed for editing vertical velocity picks, saved to an object database.

Parameters of module



The module parameters allow adjusting the display of velocity picks for further editing

Min Velocity – minimum velocity

Max Velocity – maximum velocity

Min Time – minimum time

Max Time – maximum time

Velocity scale – horizontal velocity scale in $\frac{m/s}{sm}$

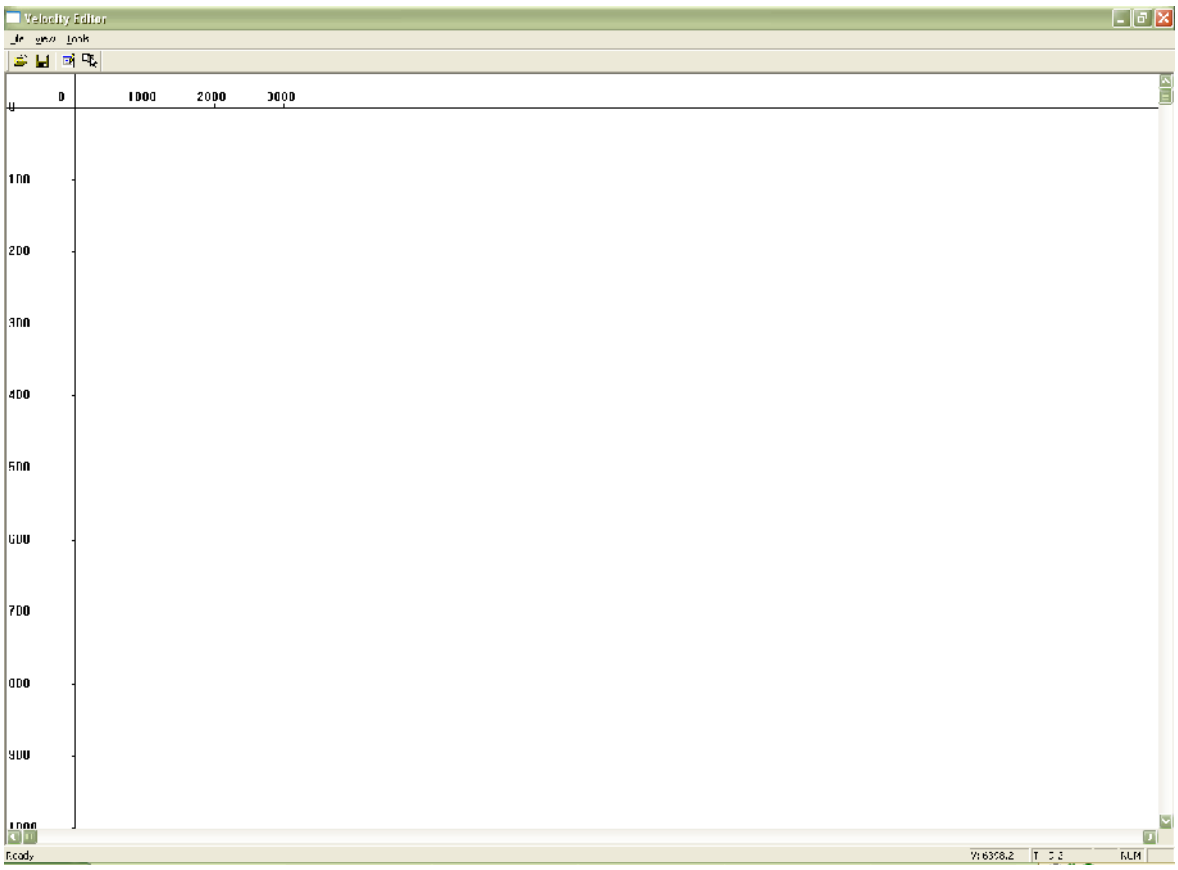
Pick to pick distance – distance between neighboring velocity picks

Save Template – saving module parameters to the database

Load Template – loading saved module parameters from the database

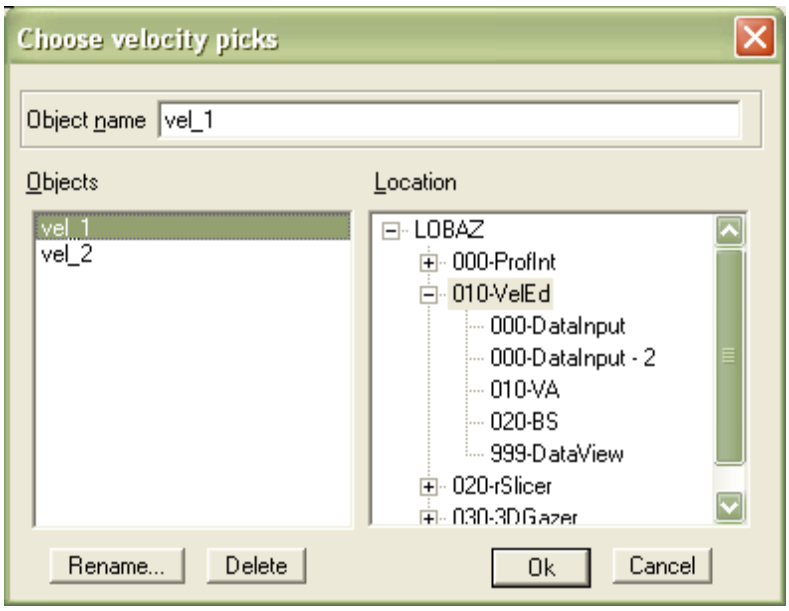
Module operation

The module is standalone, i.e. it doesn't need any additional modules in the flow, and it should be alone there. To run the module, please specify the parameters and click the button **Run**.



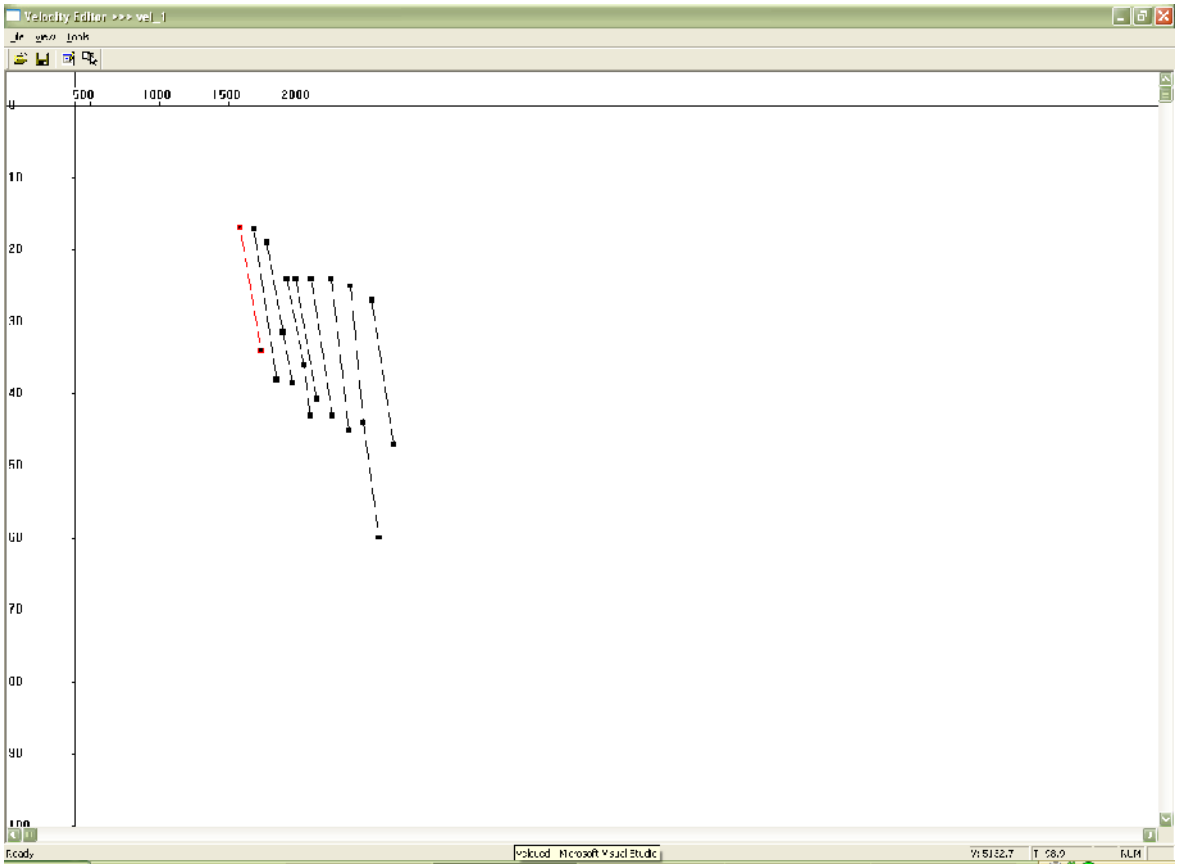
Loading velocity picks

After running the module you have to load velocity picks from the database. To do so click on the button on a toolbar, or select item **Load Velocity** from the menu **File**, or use shortcut key **Ctrl+O**. You will be invited to select the required velocity pick from the database



After selecting velocity pick, it will be displayed in the main window of the module in the form of

velocity curves' set, each of those is tied to a CDP point number.

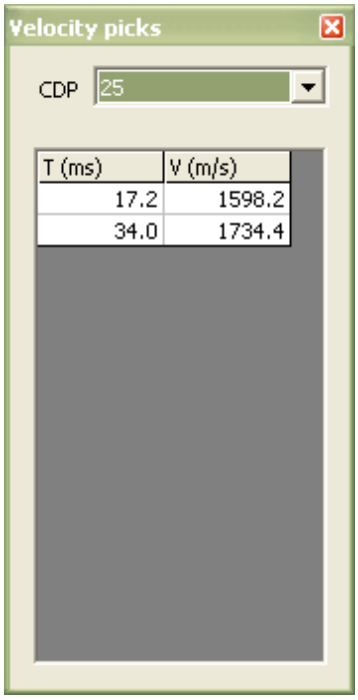


Display parameters editing

If appropriate you can modify velocity curves' display parameters, to do so you have to click on the icon , or select an item **Parameters...**from the menu **Tools**, or use shortcut key **Ctrl+P**; after that parameters' adjustment dialogue box will appear (see item **Module parameters**).

Velocity curve editing

An active velocity curve, displayed in red color is available after loading velocity pick. To change an active curve you can either **Ctrl+click** on the new curve or call a dialogue of curve editing. You can click on the icon , or select an item **Edit Velocity Picks...** from the menu **Tools** or use shortcut key **Ctrl+E.** As a result the following dialogue will appear:



Here you can select a velocity curve being edited by CDP point number (**CDP**). You can edit an active curve by dragging its nodes in the main window of the module via the left mouse button, or changing the values in the table of the dialogue manually.

Saving velocity picks

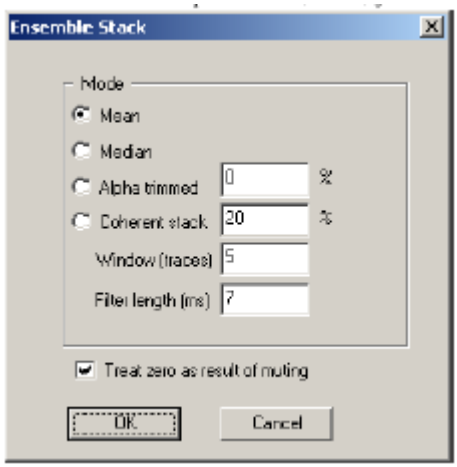
To save the velocity pick, having overwritten it, you can click the button on the tool bar or select the item **Save Velocity** from the menu **File**, or use shortcut key **Ctrl+S**. To save velocity picks under other name you should select the item **Save Velocity As...**from the menu **File**, or use shortcut key **Ctrl+Shift+S**; after that a dialogue prompting to save the velocity pick to the selected folder appears.

Stacking/Ensembles

Ensemble Stack

In the **Ensemble** stack module you can stack all traces within each ensemble of the flow into one trace. Every trace sample at the output will be a combination of corresponding trace samples at the input. An ensemble is considered as a group of traces with one and the same value of the primary sorting key defined in **Trace Input** or other input routine.

When this module is activated the following window appears:



where, in the **Mode** field,, select a trace stacking mode:

- Mean - sample values are being averaged.

- **Median** - the result sample is the median value of the samples at every trace time.

- **Alpha trimmed** - the smallest and the greatest values of the samples percentage are excluded and then the average value is defined. When selecting the **Alpha trimmed** it is required to specify the percentage of excluding.

- **Coherent stack**. The samples are stacked, if the value of the smoothed coherence function, calculated for the given time exceeds the indicated value at the given window (in percents).

- **Window (traces)** – the window of coherence function calculation is set in traces. If the value indicated in the field exceeds the number of traces in the ensemble, the whole ensemble will be used as a calculation window.

- **Filter length (ms)** – the length of time sliding window used for coherence function filtering is set in the field.

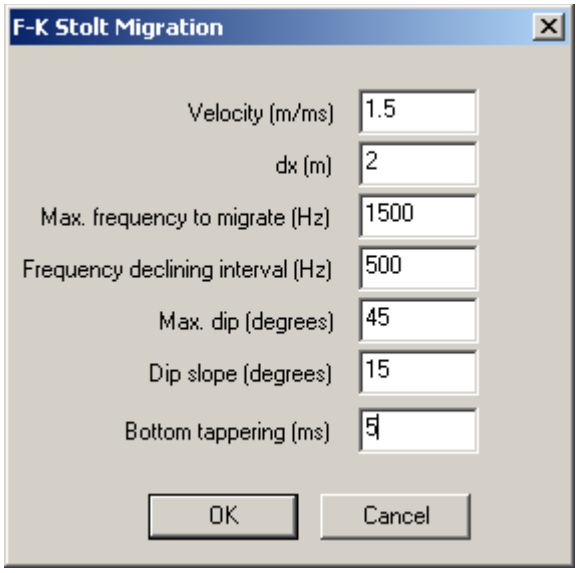
- **Treat zero as result of muting**. If this option is toggled on, the samples with zero values are treated as a result of muting and are not used in stacking.

Migration

Stolt F-K Migration

The module performs Stolt migration of seismic gathers, acquired at zero source-receiver distance. It is assumed that the traces are equidistant with the interval between the neighboring traces, assigned by **dx** parameter (in m).

Module parameters



Velocity – migration velocity (in m/ms)

Dx – distance between the neighboring traces (in m)

Max. frequency to migrate – maximum frequency (f_{MAX}) that contributes to image generation (in Hz)

Frequency declining interval – frequency declining interval Δf , the spectrum is multiplied by the function $w(f) = 0.5 + 0.5\cos\left(\pi \frac{f - f_{MAX} + \Delta f}{\Delta f}\right)$ within the range of this interval

Max. dip – maximum dip of the boundary θ_{MAX} (in degrees)

Dip. slope – range of dips, for which a smooth spectrum roll-off is to be done (Δθ).

Bottom tapering – time window at the bottom of the trace, for which a smooth bottom tapering is to be done (ms).

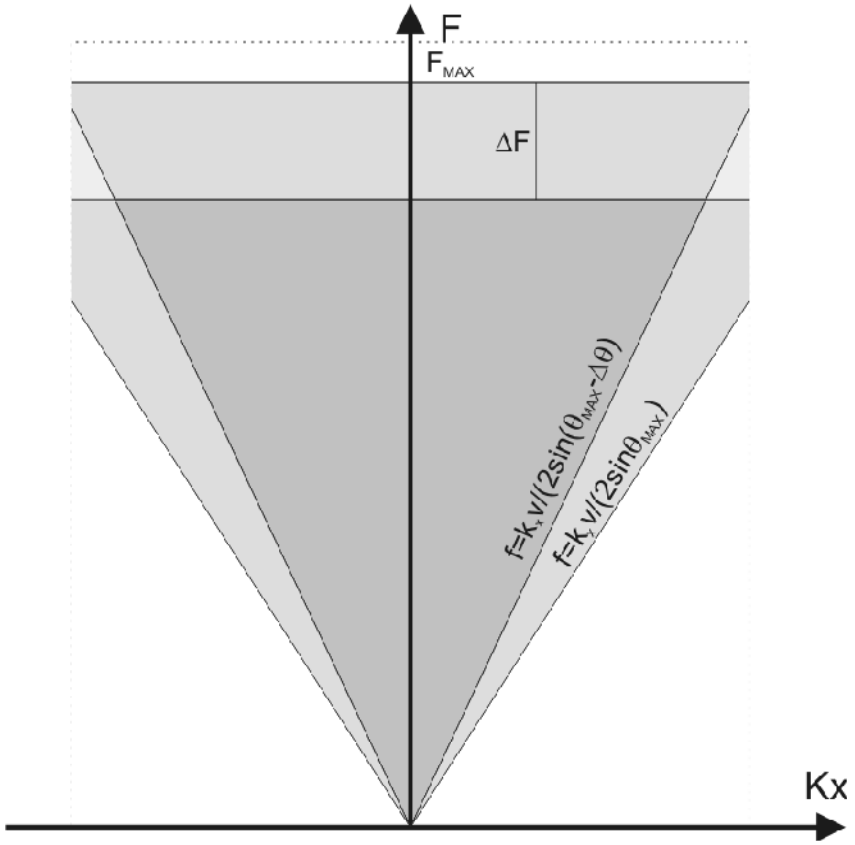
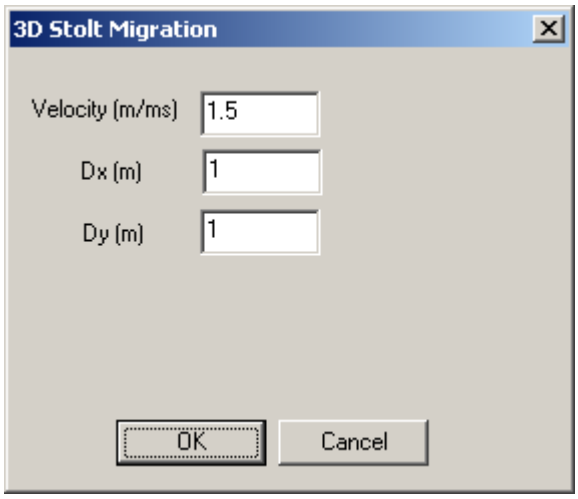


Fig. 1. Scheme of the used part of the 2D spectrum.

STOLT3D (3D Stolt F-K Migration)

The module performs 3D Stolt migration of seismic gathers, acquired at zero source-receiver distance. The traces are supposed to be located at a regular rectangular grid. If some areas of the grid contain no data, these areas shall be zero-padded. (Such data can be prepared, for instance, by the **Profile Interpolation** routine). The input data shall be sorted by ILINE_NO (in-line numbers) and XLINE_NO (cross-line numbers).

Module parameters



Velocity – migration velocity (in m/ms)

Dx – distance between the neighboring in-lines (in m).

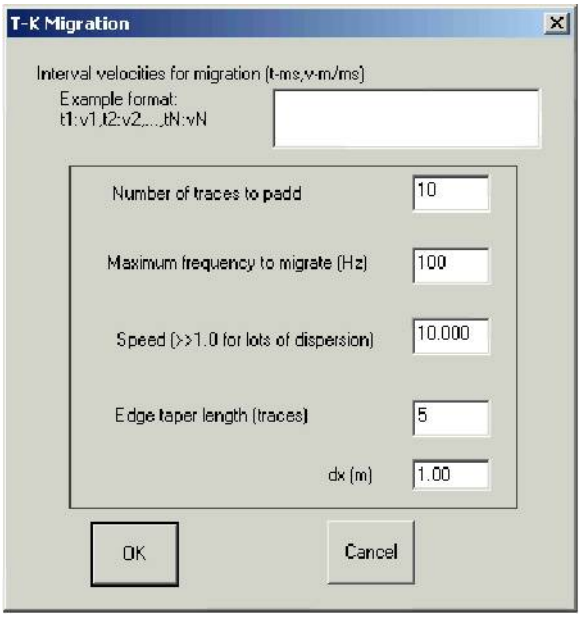
Dy – distance between the neighboring cross-lines (in m).

T-K Migration

This module is meant for T-K migration of single-channel data sorted by general offset or summed by CDP. The migration is accomplished within time-wave -number domain (T-K). The primary advantage of this method is its high accuracy in case of steep and even upset reflecting horizons and velocities varied with depth. The main disadvantage of this method is that lateral velocity variations are not taken into account. Just as in any other migration method the correct specification of migration velocities is critical. This method requires the user to specify interval velocity as a time function. It should be noted that interval velocities are usually higher than mean-root-square ones.

Parameters

When this module is activated the following window appears:



- **Interval velocities for migration** - in this field, specify the interval velocitiypairs which will be used while migration as pairs of *time: interval _velocity* values (pairs are separated by commas). Time is expressed in ms, velocity - in km/s.

- **Number of traces to pad** - here, specify the number of zero traces added to data before BPF is accomplished in order to reduce BPF cyclic effects.

- **Maximum frequency to migrate (Hz)** - maximum coherent frequency (in Hz) which is presumably contained in data. At frequencies higher than the one indicated in this field the dispersion is possible after migration. The lower the indicated value is, the higher the migration calculation speed will be.

- **Speed (>>1 for significant dispersion)** - when increasing this factor up to the value greater than one you decrease the computation time by means of increasing high frequencies dispersion for boundaries with big slopes. The value set by default (1.0) should not cause significant dispersion. This parameter is especially useful while testing various velocity functions.

- **Edge taper length (traces)** - edge taper length expressed in traces. Both edges of the section will be smoothed by weighting function in the window with specified length. The most external traces will be zero.

VSP

VSP Migration

This module is obsolete. It is recommended to use the 2D-3D VSP Migration module with similar parameters.

Curved Profile VSP Migration

This module is obsolete. It is recommended to use the 2D-3D VSP Migration module with similar parameters.

2D-3D VSP Migration

This module performs Kirchhoff migration of VSP seismograms [Dillon, 1990] and VSP-CMP transformation. The migration method is based on a horizontally layered model and does not take lateral velocity changes into account. The input of the module is a VSP seismogram passing through the flow with separated nonconverted reflected wave field and completed DEPTH, REC_X, REC_Y, REC_ELEV, SOU_X, SOU_Y, SOU_ELEV header fields.

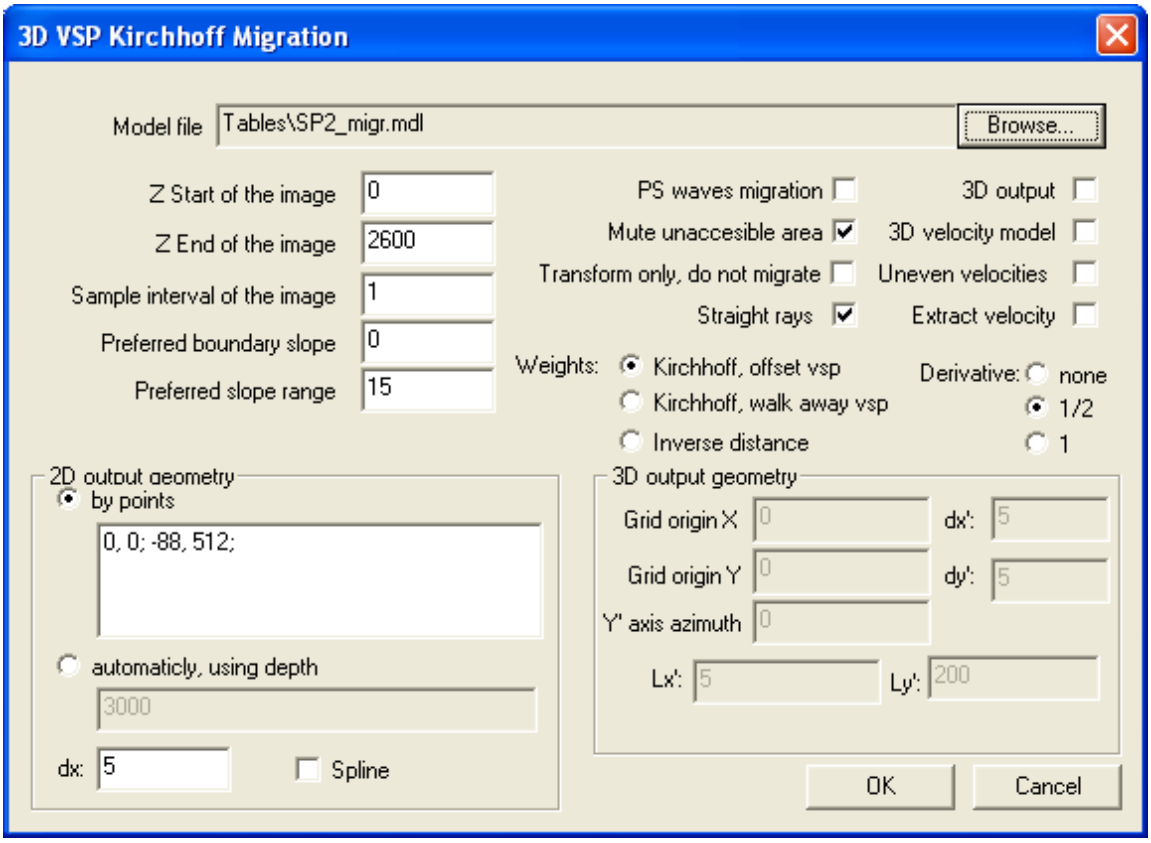
The following migration strategy is employed:

- A layered velocity model with sufficient layer thickness (when 10-meter observation intervals are used, introduction of layers less than 50-60 m thick into the model is justified only in certain cases) is built based on the near shot point data. In some cases, vertical anisotropy may be taken into account.

- A velocity model is selected in the space below the well bottom based on prior data.

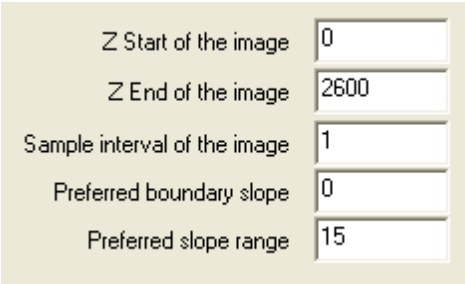
- After the nonconvereted reflected wave field of the near or far shot point is cleared of interference and noise, it is used to obtain a migrated profile.

The module parameter dialog box looks like the following:



Model file – this field allows selecting a migration velocity model file. Such file may be generated, for example, by building a velocity model in the **Advanced VSP Display** module (see the description of this module for the file format)**.**

Click the **Browse…**button to select a file.

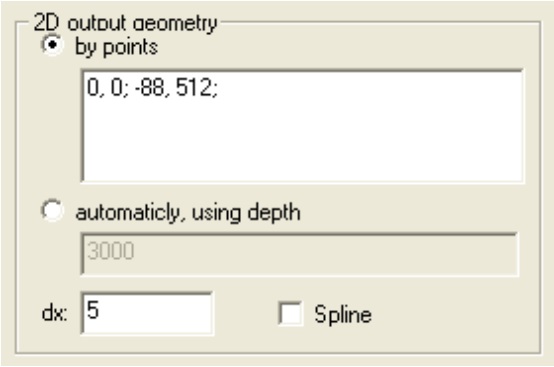


Z Start of the image, Z End of the image – start and end depths (m) of the migrated section.

Sample interval of the image – sample interval (m) of the migrated section.

Preferred boundary slope, Preferred slope range – these two parameters allow adjusting the filter for the expected dip moveout and specifying the expected angle and angle range (in degrees), respectively.

2D output geometry – a set of parameters used to define the migration line.



This box becomes active when the **3D output** option is disabled.

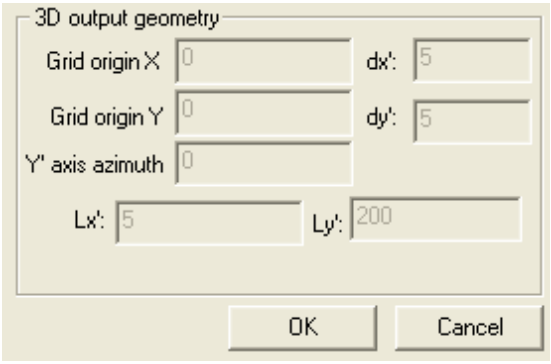
- **by points** – the line is defined using points, i.e. sections with coordinates specified interactively in meters in the X1, Y1; X2, Y2 format.

- **automatically, using depth** (m) – the migration line is defined automatically. The maximum depth is entered into the box, and the migration line is calculated relative to the first source position in the specified depth range (enabling this option is not recommended when working with data obtained using the "Walk-Away VSP" method).

- **dx** – interval along the profile in meters.

- **Spline** – use/do not use spline smoothing.

3D Output geometry – a set of parameters used to define the 3D migration grid.



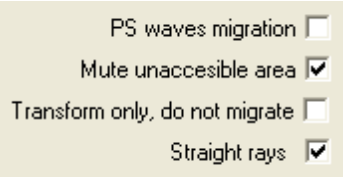
This option is active only when **3D output** is enabled**.**

Grid origin X, Y – grid origin coordinates in meters.

Y` axis azimuth – clockwise grid rotation angle relative to the Y axis azimuth in degrees.

dx`, dy` – grid spacing in meters on the X and Y axes, respectively.

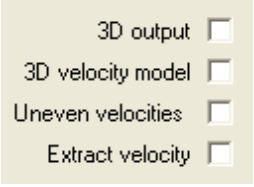
Lx`, Ly` – grid size in meters on the X and Y axes, respectively.



PS Wave migration – converted PS wave migration. This box is active when the **3D output** option is disabled.

Mute unaccessible area – muting of seismogram areas corresponding to "invisible" (within the horizontally layered model and the known observation geometry) medium areas. This box is active when **3D Output** is disabled.

Transform only, do not migrate – if this option is enabled, the module performs VSP-CMP transformation; if it is disabled, the module performs migration.



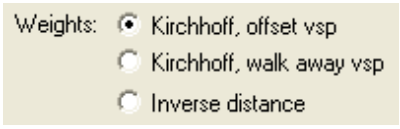
Straight rays – if this option is enabled, migration will be performed in the time domain, and interval velocities will be converted to RMS velocities.**3D Velocity model** – read/do not read the

text file containing the velocity model for migration from the 3D format (format description is provided below). This option is active only when **3D output** is enabled.

Uneven velocities – if this option is enabled, different velocities will be used for rays going from the source to the image point and from the image point to the receiver in the process of migration. The velocity function for each point is defined based on the specified **3D velocity model** as a value corresponding to the middle of the image – source and image – receiver distance. If this option is disabled, the same velocity function will be used for every image point with the same horizontal position. This option is active only when **3D output** is enabled.

Extract velocity – if this option is active, an interval velocity field will be displayed in the migration result visualization window instead of the migrated image (this option is still under development).

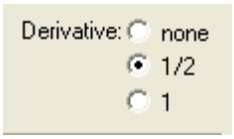
Weights – use of different weights in Kirchhoff integral for OVSP (Kirchgoff, offset vsp) and



Walk-Away VSP (Kirchgoff, walk away vsp).

The **Inverse distance** option allows using values inverse to distances as weights.

Derivative – settings for the filter applied to the data before migration taking the wavelet shaping factor (**none, 1/2, 1**) into account.



Wavelet shaping for 2D Kirchhoff migration is determined by a 45-degree constant-phase spectrum and an amplitude spectrum proportional to the square root of the 2D migration frequency (**Derivative = 1/2**).

For 3D Kirchhoff migration, the phase displacement is equal to 90 degrees, and the amplitude is proportional to the frequency (**Derivative = 1**).

For VSP-CMP transformation, the phase displacement is equal to 0 degrees (**Derivative = none**).

NOTE 2. 3D Velocity model file format description.

The file should contain interval velocities with [km/s] dimensionality. The velocity function should be defined for each node of the regular 2D grid. Each velocity function may contain an arbitrary number of layers with known base depths (in meters) and velocities for each of them as well as a column with longitudinal wave velocities (in km/s).

The module reads the velocity field from a text file with the following structure:

1st line should always begin with #VSP and may contain any comments

2nd line: the number of cells into which the X axis will be divided, and interval on the X axis in meters. Comments may be added after a # symbol.

3rd line: the number of cells into which the Y axis will be divided, and interval on the Y axis in meters. Comments may be added after a # symbol.

4th line: X coordinate of the grid origin, Y coordinate of the grid origin, and Y axis azimuth.

Starting from the 5th line, all lines contain the velocity function defined for the set of layers. Each line consists of layer base depth, velocity, total number of layers in this velocity function, number of cells on the X axis, and number of cells on the Y axis.

An example of 3D Velocity model file structure:

#VSP migration velocity model file, generated by CVelocityCube

2 10000.000 #NX, DX

2 10000.000 #NY, DY

0.000 0.000 0.000 #X_origin, Y_origin, Y_azimuth

1410 2.0 2 0 0

4010 2.1 2 0 0

1410 2.0 2 0 1

4010 2.1 2 0 1

1410 2.0 2 1 0

4010 2.1 2 1 0

1410 2.0 2 1 1

4010 2.1 2 1 1

The module produces a migrated section or a VSP-CMP section plus a one-dimensional velocity model (distribution of velocities along the profile for a single depth or along the depth for a single image position on the axis).

Comments on use

Regardless of selected parameters, the medium is assumed to be one-dimensional. It means that ray tracing is performed for a horizontally-layered medium, and the method of migration in the time domain assumes the medium to be one-dimensional for each ray (however, different rays may have different velocity functions).

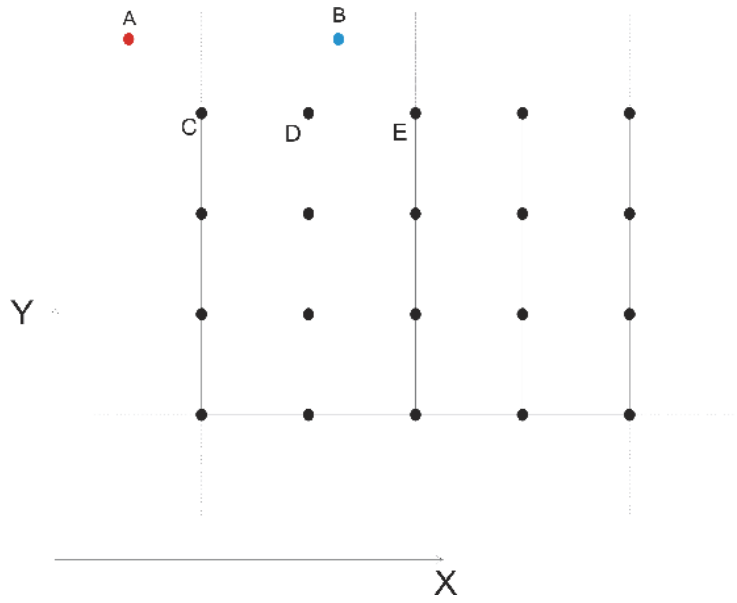
Obtaining a one-dimensional velocity function from a 3D velocity cube.

A velocity function should be defined for every node of the regular 2D grid. The following approach is used to obtain velocity functions for points with specified coordinates:

1) If the point for which velocity is to be determined falls outside the grid and is not located between any two lines serving as grid line extensions (point A in the figure below), the velocity value for that point is selected equal to the nearest grid node (in this case – point C in the figure).

2) If the point for which velocity is to be determined falls outside the grid and is located between any two lines serving as grid line extensions (point B in the figure), the velocity value for that point is determined using the values in the nearest 2 nodes (in this case – points D and E in the figure).

3) If the point for which velocity is to be determined is located inside the grid (and, therefore, inside a certain cell), the velocity value for that point is determined using the values in the nodes of the cell containing the point in question.



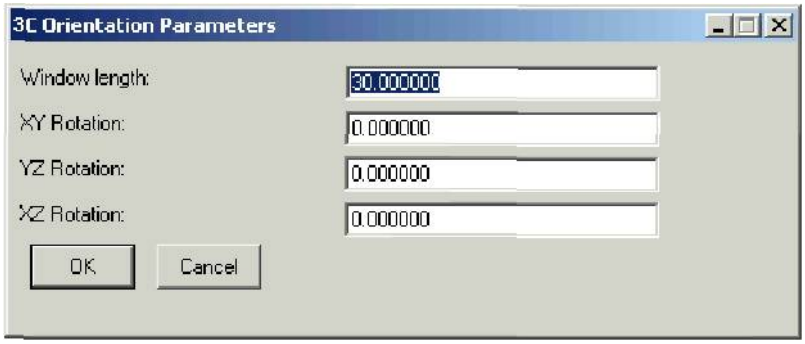
If the velocity value for a given point is determined using more than one velocity function, the following principles apply:

First all velocity functions are normalized for the number of layers in the model. This is achieved by determining velocity values at layer boundaries and then applying linear or bilinear interpolation to determine velocity values for each point.

3C Orientation

This module converts PM-VSP seismograms into PRT system by means of P-component orientation on the energy maximum in the window.

Parameters



- **Window length** - the length of window from first arrivals (expressed in ms) in which the seismogram orientation will be carried out according to data in this window. If the window is too small it may cause unstable routine operation. If the window is too big besides direct P-wave, the waves reflected from adjacent boundaries, refracted on them with conversion and other waves will get in it.

- **XY Rotation, YZ Rotation, ZX Rotation** - these parameters (expressed in degrees) allow additional rotation of the coordinate system in respective directions.

Data preparation

The data on first arrival times this module obtains from FBPICK field of traces headers. That is why, before this module application you write them in. To do this you should:

- Select the **Depth, Depth** fields in the **Tools/Pick/Pick Headers** menu. Then, define the times of first arrivals and save them as a pick.

- Start visualization of field data of all three components by indicating the CHAN, DEPTH (it is better to specify them in this order though it is not compulsory) header fields in the **Sort field** of the *Trace* ***Input*** and "*:*" in the string editor below **Sort fields.** It is as well useful, though it is not compulsory, to select the **Ensemble boundaries** flag in the parameters of visualization.

- Load first arrivals pick **(Tools/Pick/Load Pick).**

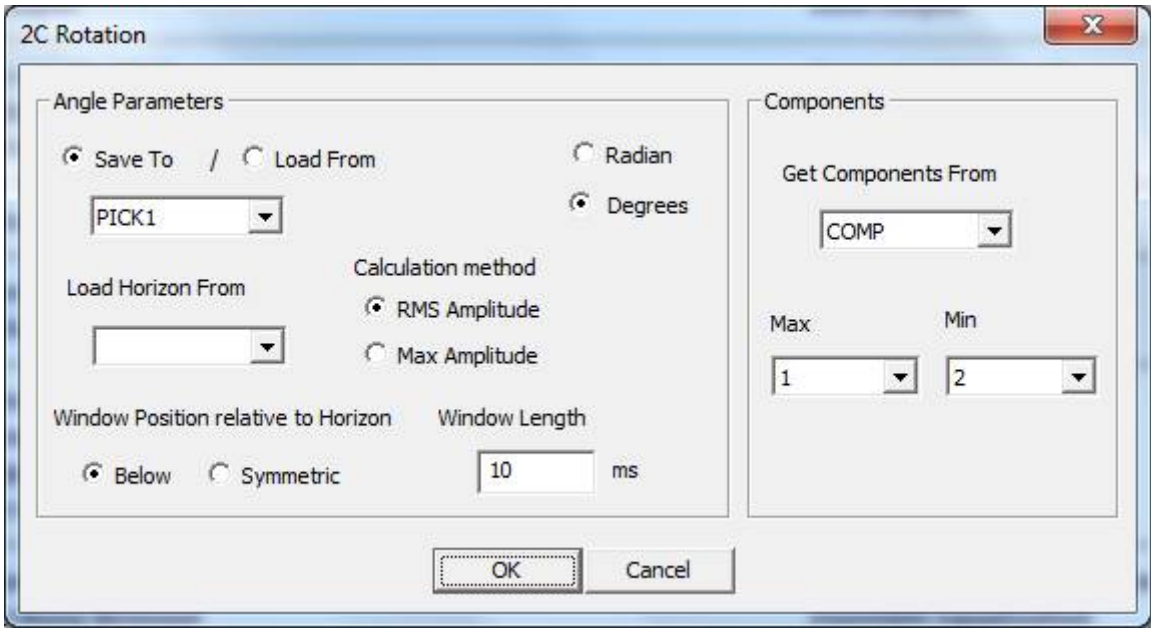
- Save the pick to file headers **(Tools/Pick/Save to headers).**

After the pick has been saved to file headers the corresponding file should be saved and registered

in the database. Beside, the coordinates of source and receiver positions should be saved to headers.

In order to enable module operation the input data must be sorted so that the traces corresponding to one depth were alongside. This can be. For example, achieved by specifying the CHAN, DEPTH (this order is compulsory) header fields in the **Sort field** field of the *Trace* ***Input*** and "*:*" in the string editor below **Sort fields.**

2C Rotation



This module is used for orientation of any two components of multicomponent VSP data.

Data to be input into the module may be either 2C or 3C and must be sorted by depth and component number. The components must be numbered as 1, 2 and 3.

The header field containing the component numbers may be selected using the **Get Components From** option (COMP header field is set by default).

As a result of running the module, two components specified in the **Max** and **Min** drop-down lists are rotated according to the calculated values while the third component (if any) remains unchanged.

The amplitude values used for rotation angle evaluation are calculated within the window specified in the **Window Length** field in relation to the horizon pick that must be stored in a header field that is specified in the **Load horizon from** drop-down list. The window can either be **Symmetric** relative to the pick, or **Below** the pick.

The module employs two methods to determine the component rotation angle (**Calculation method** field):

1. ***RMS Amplitude*** – root-mean-square amplitudes are calculated over the specified window

for the two components selected. These two values of RMS amplitudes are taken as a projection of the full amplitude vector and are used to build the vector. Then the component selected in the **Max** field is oriented along this vector while the component selected in the **Min** field is oriented perpendicular to the vector.

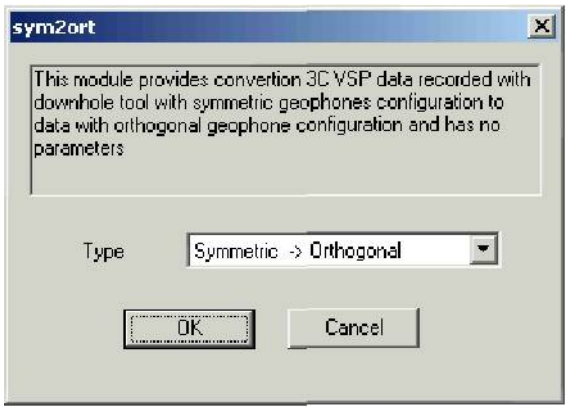
2. ***MAX Amplitude*** – a set of vectors based on individual pairs of amplitude samples at two components within the window is built. The vector with the maximum length is assumed as the orientation of the component selected in the **Max** field. The component selected in the **Min** field is oriented perpendicular to the longest vector.

Save to – if this option is selected, the module outputs the components rotated at the calculated angle and saves the angle to the corresponding header.

Load from – if this option is selected, the module does not calculate angles bu simply take them from a specified header field and rotates the components selected in the **Min** and **Max** fields accordingly.

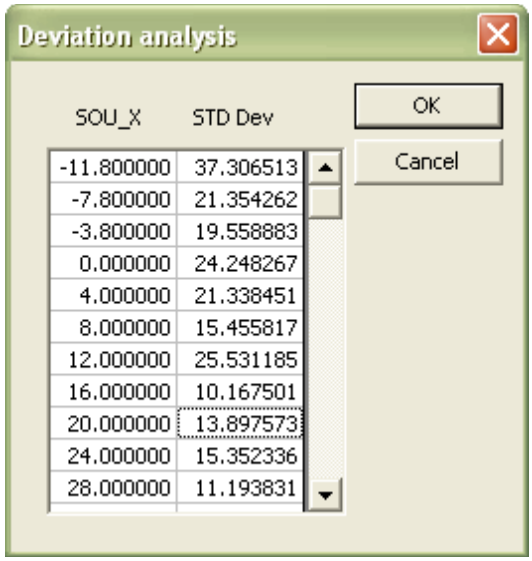
sym2ort

This module is meant for conversion of 3C data recorded with downhole tool with orthogonal geophones configuration to data recorded with downhole tool with symmetric geophones configuration and vice versa.


sym2ort
This module provides convertion 3C VSP data recorded with downhole tool with symmetric geophones configuration to data with orthogonal geophone configuration and has no parameters
Type
Symmetric -> Orthogonal
OK
Cancel

The only parameter of the module **(Type)** defines the direction of conversion.

The input data should be sorted by receiver location and, after that, by components (for example, DEPTH:CHAN - by cable depth and, after that, by channel). It is assumed that the CHAN field contains the record component (1 - X, 2 - Y, 3 - Z or 1,2 or 3 for symmetric configuration). Besides, it is always assumed that one receiver location is corresponded by 3 record components.



VSP Geometry*

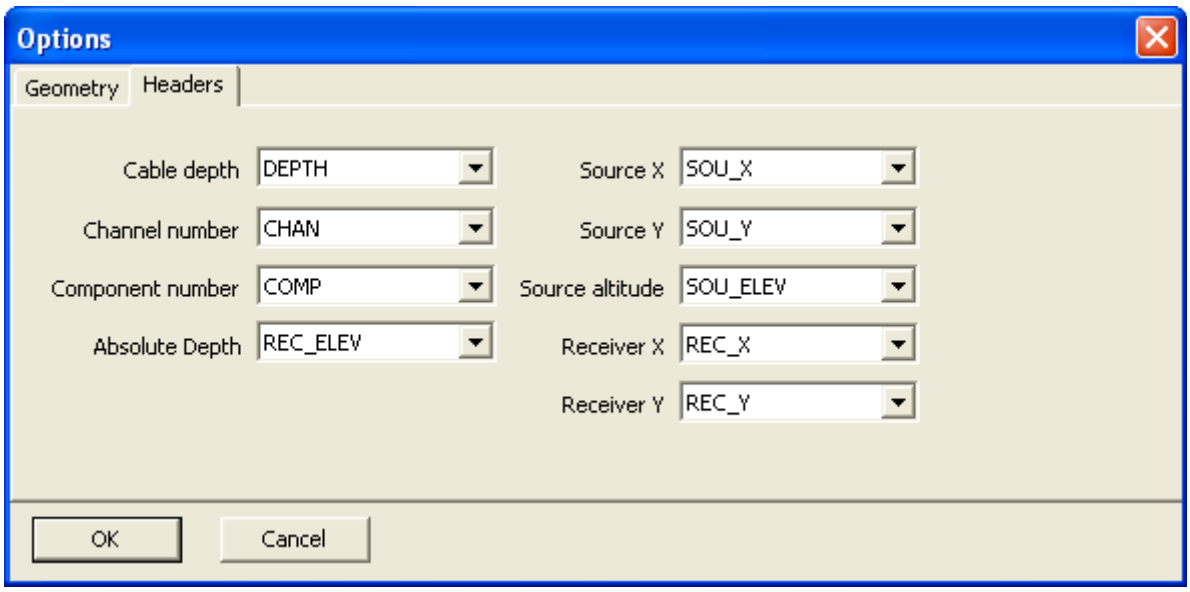
The module is designed for input of geometry and inclinometry into VSP data. The **VSP Geometry*** is a stand-alone module, that is it does not need any additional input/output routines to be in the flow. The dataset in the project database where the geometry is to be input to is a parameter of the module itself.

The module fills in a standard set of trace header fields needed for VSP processing, as well as input inclinometry information when needed.

Module parameters

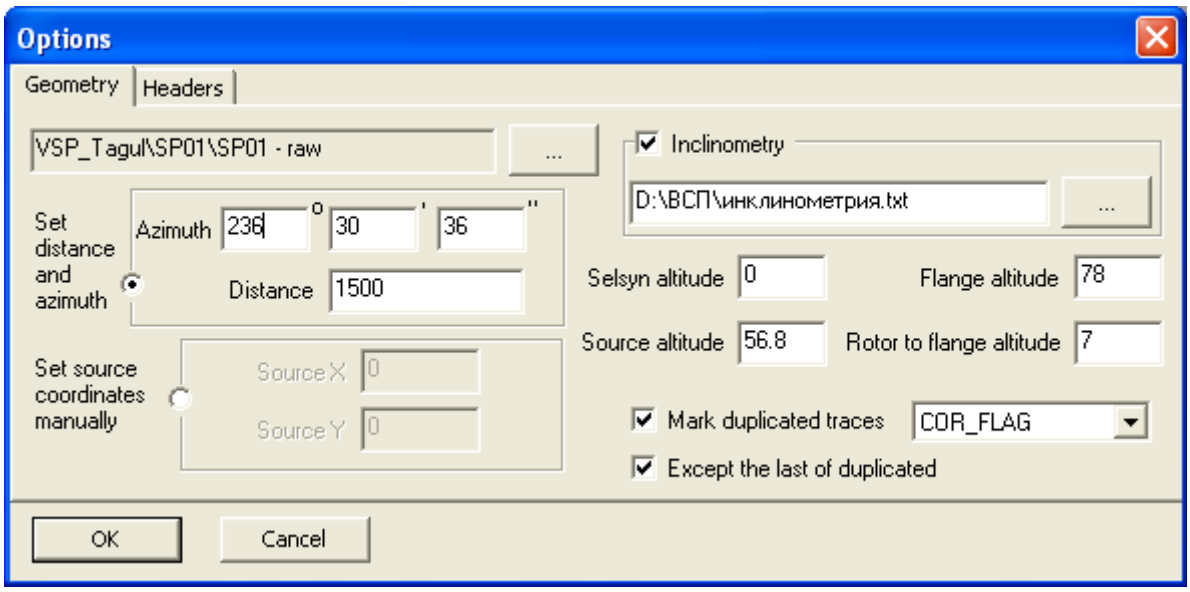
The parameter dialog of the routine contains two tabs: **Geometry** and **Headers.**

On the Headers tab specify trace header fields to be read or overwritten during the module operation.

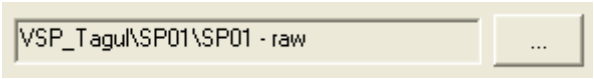


- ***Cable depth*** – this is an *input* field, it shall be properly filled in beforehand.
- ***Channel number*** – this is an *input* field, it shall be properly filled in beforehand.
- ***Component number*** – this is an *input* field, it shall be properly filled in beforehand.
- ***Absolute Depth*** – this is an *output* field, it will be overwritten by the module.
- ***Source X*** — this is an *output* field, it will be overwritten by the module.
- ***Source Y*** - this is an *output* field, it will be overwritten by the module.
- ***Source altitude*** – this is an *input* field, it shall be properly filled in beforehand.
- ***Receiver X*** - this is an *output* field, it will be overwritten by the module.
- ***Receiver Y*** - this is an *output* field, it will be overwritten by the module.

On the **Geometry** tab specify the following:



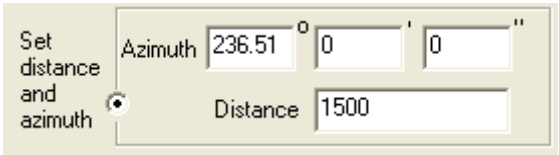
Click the button to select the dataset name where the geometry is to be input in. When a dataset is selected, its name will be displayed to the left of the button:



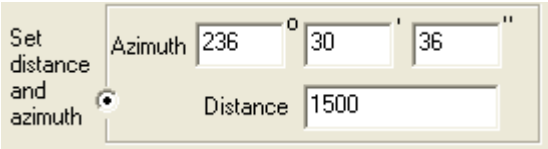
Select one of the ways how source coordinates are to be evaluated:

Set distance and azimuth – the source X and Y coordinates will be calculated basing on its azimuth and distance from the collar of the well (the coordinates of the collar of the well are considered to be 0,0):

Azimuth – defined in degrees, minutes and seconds from the north. Any of the three fields can be decimal. After the dialog is closed and reopened again, the specified azimuth will be recalculated into the form of degrees, minutes and seconds with decimals. For instance, you can specify the azimuth as following:

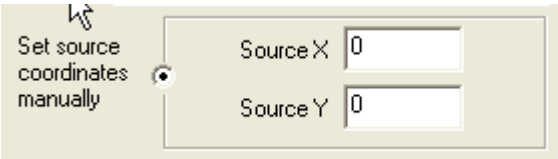


After the dialog is reopened, it will be represented as shown below:



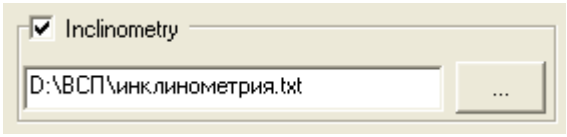
Distance – defined in meters from the well collar.

Another option is to ***Set source coordinates manually***.



Specify ***Source X*** and ***Source Y*** in meters.

Inclinometry – this flag allows loading inlinometry information for an inclined well.



If it is checked, the REC_X and REC_Y values will be calculated basing on the inlinometry information read from a specified file (the file format is described in the Appendix 1 below):

Otherwise, REC_X and REC_Y will be set to 0.

Other parameters are:

- **Selsyn altitude** – specified in meters.
- **Source altitude** – specified in meters.
- **Flange altitude** – specified in meters.
- **Rotor to flange altitude** – specified in meters.

(See scratch in the Appendix 2 for explanation).

Mark duplicated traces – if this flag is checked, the traces with the same value of cable depth (DEPTH) and of the same component (COMP) will be marked, by 1 written to the user-specified header filled. For all other traces, this header will be assigned 0.

Except the last of duplicated flag is active only when the ***Mark duplicated traces*** is checked. When this flag is checked, the last one of the duplicated traces is not marked – its marking header is assigned 0. This ensures that at least one trace of each set of duplicated traces is not marked, and

thus can be easily kept while the others can be sorted out on input.

Module operations

The module use the following formulas:

- For source position (if not assigned explicitly):

Source X= Distance * Sin (Azimuth/180*Pi)

Source Y= Distance * Cos (Azimuth/180*Pi)

- For receiver positions:

Without inclinometry:

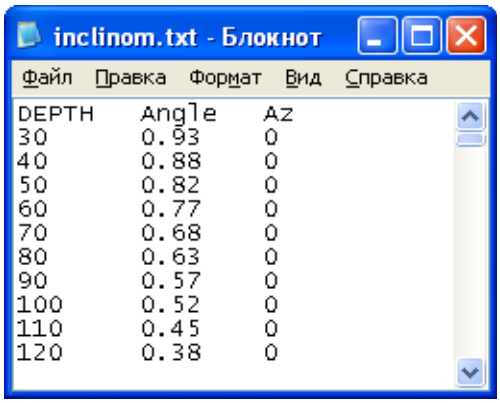
Absolute Depth = Cable depth - Rotor to flange altitude

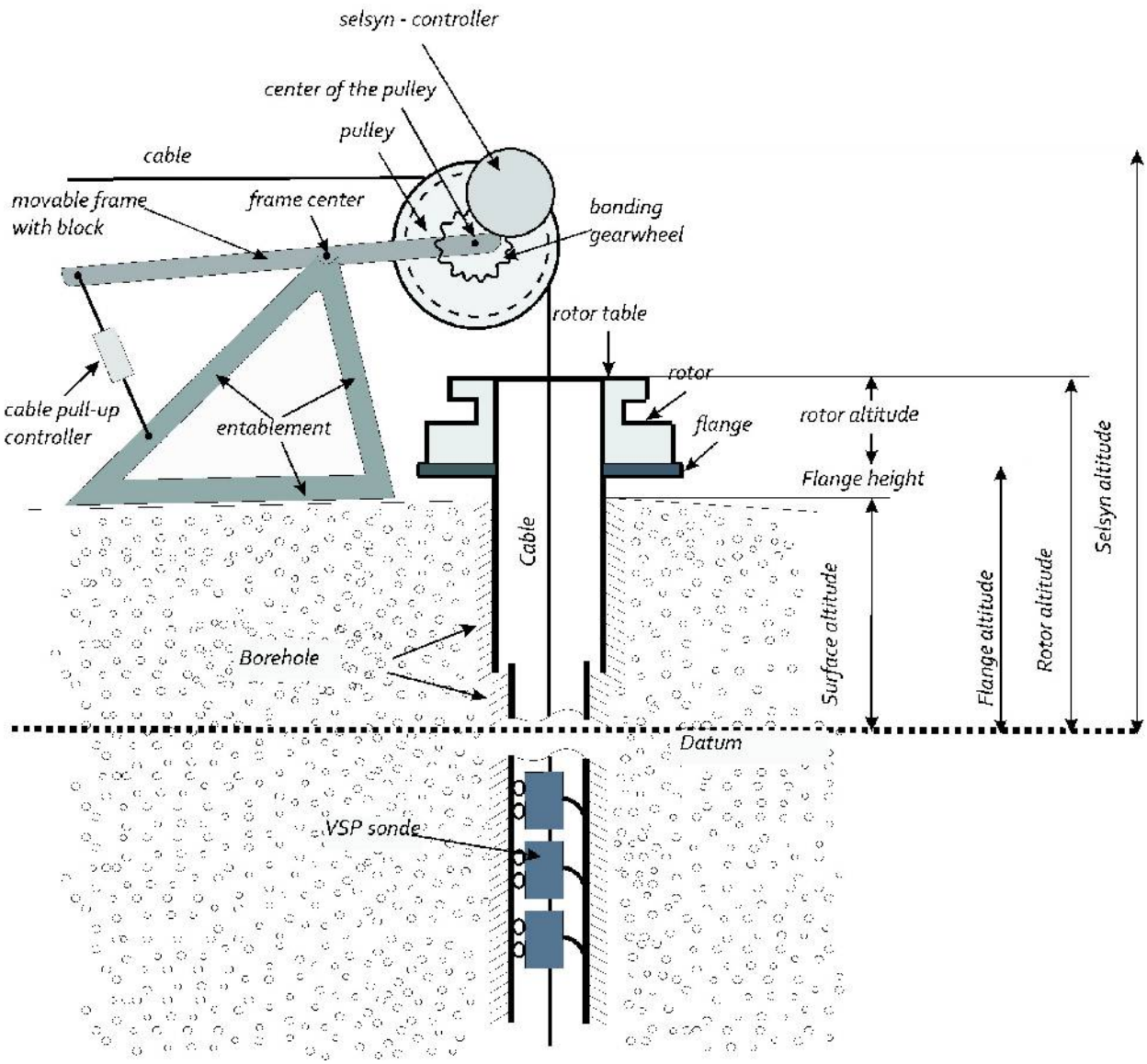
Receiver X = Receiver Y = 0

- With inclinometry, the absolute depth, X and Y of the the receivers are calculated basing on the interpolated inclinometry information/

Appendix 1. Inclinometry file format.

It is a tabulated ASCII file with 3 tab or space separated columns: cable depth in meters (DEPTH), vertical angle in degrees (Andgle), azimuth in degrees (Az). An example of the file is shown below:



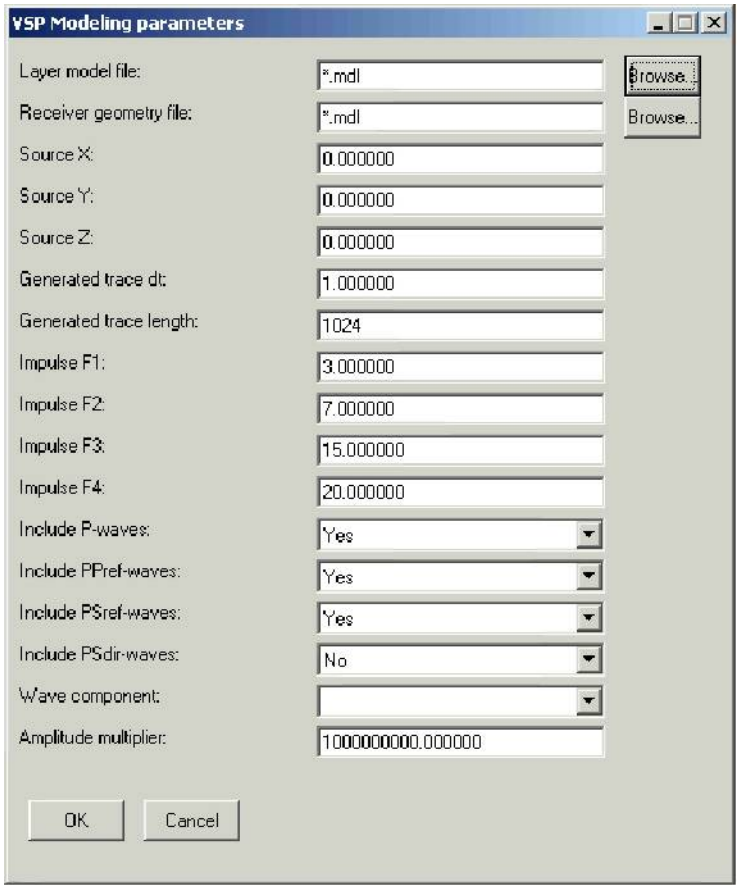


VSP Data Modeling

This module allows you to create a synthetic gather of transmitted P-waves , refracted converted PS-waves , reflected non-converted PP-waves , and reflected converted PS-waves , and enter it into the flow. The problem is solved for a horizontal-layered model at arbitrary position of source and

receivers.

When the module is activated the following dialog box appears:



Parameters

Layer model file is a velocity model file. File selection is accomplished in a standard Windows dialog box after clicking the **Browse...** button. This should be a text file with spaces or tabs used as separators. It must contain a column for the depths of the bottoms of all layers (depth should be expressed in meters) and a compressional wave velocity column (velocity should be expressed in km/sec). If shear waves are to be taken into consideration there should also be a shear wave velocity column. In the first line of the table the names of columns should follow after the "~A" symbols. The depths will be read from the Z column, compressional wave velocity from the Vlay column, and shear wave velocity from the Vslay column.

Here is an example of such a table:

~A	Z	Vlay	Vslay

10	1.6	0.8	
2000	3.2	1.791	
2540	2.6	1.4	
3000	3.51	1.54	

Receiver geometry file is a file containing the data on sources' location (distance, expressed in meters), the file structure is similar to that in previous cases but the following names of columns can be distinguished:

- *DEPTH:* cable depth
- *X* – source: X coordinate
- *Y* – source: Y coordinate
- *Z* – source: Z coordinate

While calculating, the DEPTH is not taken into account but it is simply entered into the headers of traces under creation.

Source X – source X coordinate (m). **Source Y –** source Y coordinate (m). **Source Z –** source Z coordinate (m).

Generated trace dt - sampling interval of generated traces (msec).

Generated trace length - length of generated traces (msec).

Impulse F1, Impulse F2, Impulse F3, Impulse F4 - define amplitude frequency spectrum of the source wavelet to be used for generation of synthetic traces. The spectrum is built according to the following rules:

$$A(f) = \begin{cases} 0, f \in (0;F1) \cup (F4;\infty) \\ 1, f \in (F2;F3) \\ \cos\left(\left(1 - \frac{f-F1}{F2-F1}\right) \times \frac{\pi}{2}\right), f \in (F1;F2) \\ \cos\left(\frac{f-F3}{F4-F3} \times \frac{\pi}{2}\right), f \in (F3;F4) \end{cases}$$

Generated wavelet is zero-phase.

Include P-waves - include/do not include transmitted P-waves

Include PPref-waves - include/do not include reflected non-converted PP-waves

Include PSref-waves - include/do not include reflected converted PS-waves

Include PSdir-waves - include/do not include transmitted converted PS-waves

Wave component - two options are available: X-component and Z-component

Amplitude multiplier - all traces are multiplied by this amplitude multiplier, which is 1,000,000,000 by default. This parameter does not have any physical nature.

VSP SDC (Vertical Seismic Profiling Spherical Divergence Correction for non-converted reflected compressional waves)

Theory

It is common knowledge that as a result of front divergence the spherical wave amplitudes decrease as 1/R, where R is the distance traversed by wave. As a rule, it is supposed that the distance is proportional to the wave traveltime, which is true for homogeneous medium. While vertical seismic profiling data processing, as supposed for horizontal layer model, it is possible to define velocity law and use it for more precise determination of divergence influence on reflected wave amplitude.

For the nearest shotpoint (SP) you can use an incident wave arrival-time curve. Let us to assume that

for some trace the time of arrival of direct incident wave equals T1 and the time of arrival of the wave reflected from some boundary equals T_2. Then, in case when SP is close enough to the wellhead, the time, when the wave falls on the boundary equals $T3=T1+(T_2-T1)/2=(T_2+T1)/2$. Thus, by using the incident wave arrival-time curve (having extrapolated it below the borehole bottom either linearly or with the help of information about velocity law) you can define the depth of this boundary and, of course, the reflected wave path to the boundary and after reflection: $R=R_{\text{before falling on the boundary}}+R_{\text{after reflection}}$.

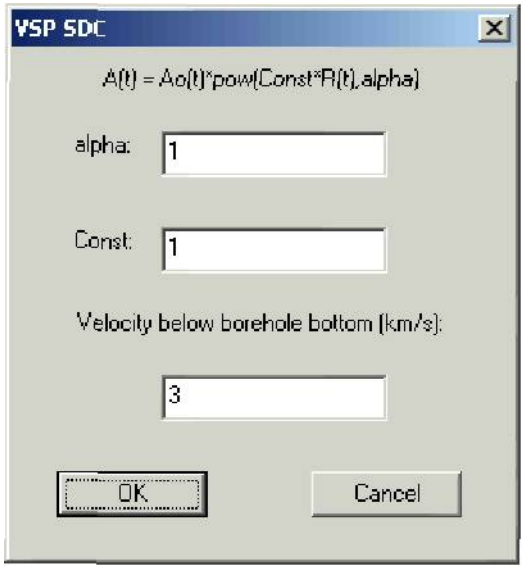
Input data

The input data for vertical seismic profiling spherical divergence correction is a seismogram of compressional VSP with filled in header fields:

REC_ELEV, SOU_ELEV - coordinates of receiver/source FBPICK - first-arrival time.

Parameters

When this module is activated the following window appears:



- **Alpha** - the degree for R(t) function (see Algorithm), set 1 by default

- **Const** - constant multiplied on correction function, set 1 by default

- **Velocity below borehole bottom** - velocity below the borehole bottom (expressed in km/s), set equal 3 km/s by default

Algorithm

- Pairs of REC_ELEV/FBPICK trace header values are selected for all traces within the whole frame. Further, these pairs are sorted according to REC_ELEV. The repeated depths are processed, i.e. if there are several traces with the same REC_ELEV value then in this case the FBPICK is assumed as mean.

- For every trace the following procedures to be applied:

- The following formula to be calculated

$$F(t) = \begin{cases} t < FBPICK; (R(FBPICK) \times Const)^a \\ t \le FBPICK; (R(t) \times Const)^a \end{cases}$$

Here:

$R(t)=Z_{refl}*2-Z_{rec}$,

where Z_{rec}=REC_ELEV-SOU_ELEV, and Zrefl is definedas following:

- If t1=(t+FBPICK)/2<$FBPICK_Z max$ (the first-arrival time on the deepest trace), then Z_{refl} is calculated from interpolated arrival-time curve.

- If t1=(t+FBPICK)/2>$FBPICK_Z max$, then it is defined by extrapolation of arrival-time curve below the borehole bottom according to specified apparent velocity V from the following expression:

$Zrefl=(t- FBPICK_Z max)*V+Zmax$

where FBPICKZmax is a maximum FBPICK value for the whole frame.

- After that, the initial trace $A_0(t)$ is multiplied by F(t), i.e. $A(t)=A_0(t)*F(t)$.

Advanced VSP Display

This module is meant for interactive joint VSP and geophysical survey data interpretation in order to create the layer velocity model, to define the quality of the medium and the VSP arrival-time curve.

The module interface consists of two parts: a window for initial parameters setting and a primary running window that appears during execution of this module. The dialog box for parameters setting can be opened during module selection, but, in this case, all parameter changes will be active only during the current module execution session.

Initial data

The initial data for the module should be organized in the following way:

VSP seismogram with filled in fields: REC X, REC Y, REC_ELEV, DEPTH (coordinate and receiver cable depth), SOU_X, SOU_Y, SOU_ELEV (source position data), FBPICK - transmitted wave arrival time. Incident wave amplitude (without amplitude divergence correction, etc., but after taking into account the possible increasing gain factor of recording equipment) calculated via the SSAA module can be contained in a separate field. The seismogram should be sorted according to increasing depth and should not contain traces with repeating receiver depth.

Well-logging data can be transmitted in an ASCII text table file with the following structure: *~A*

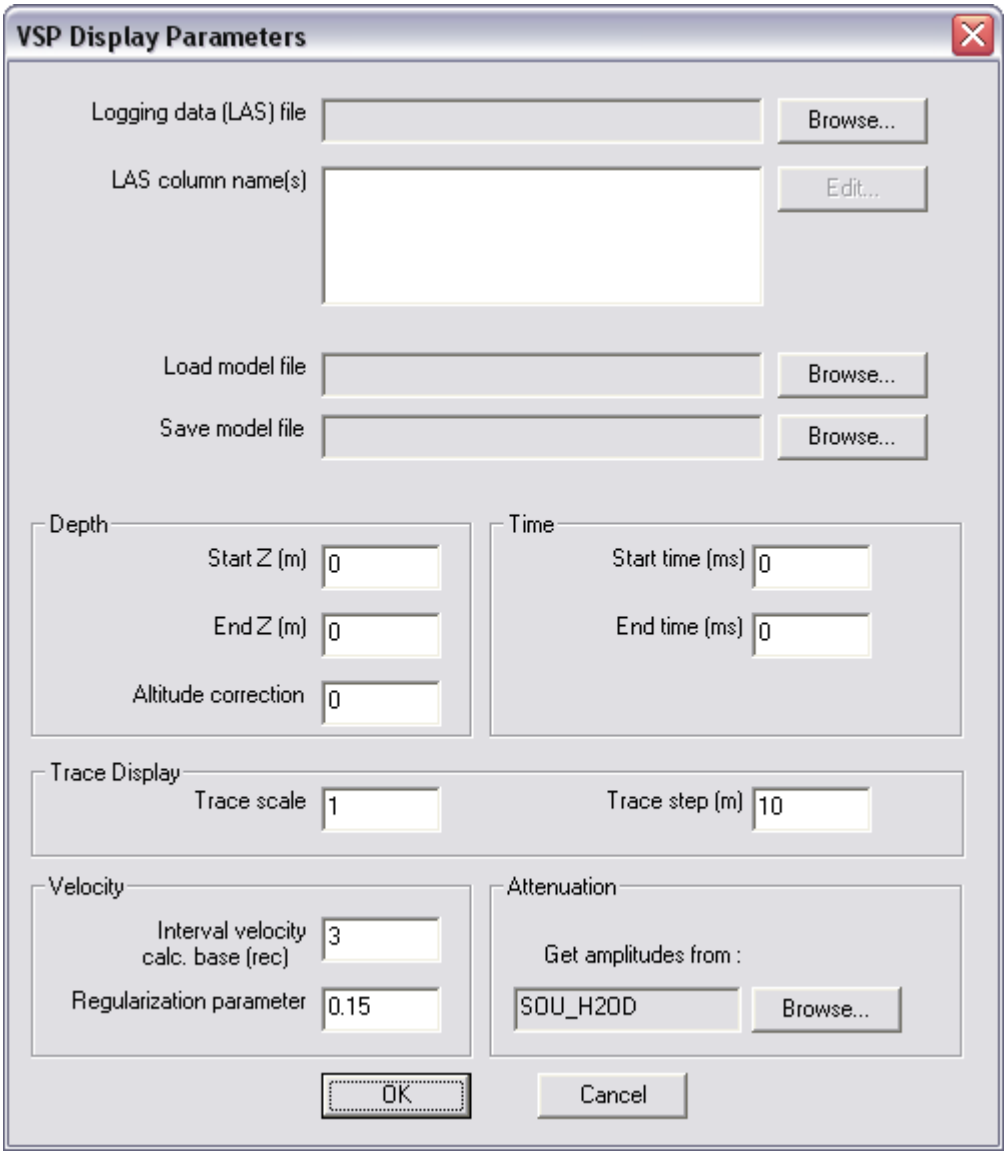
DEPTH N1 N2 N3

100 1.1 2.1 3.1

200 1.2 2.2 3.2

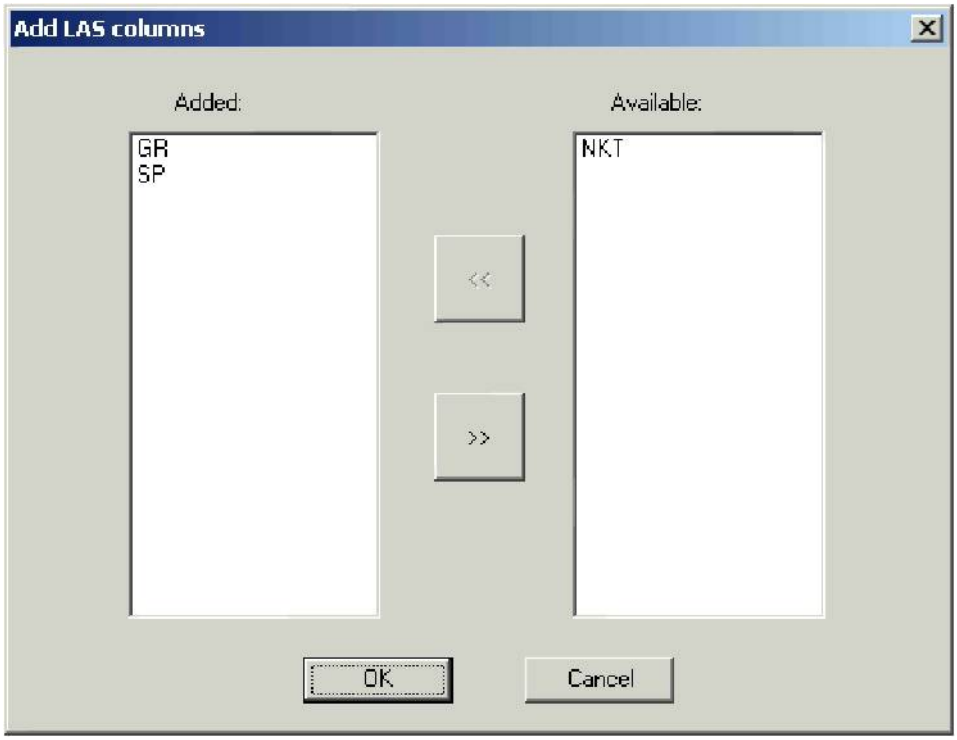
In the DEPTH column, the cable depth is indicated. In the rest of the columns the data of some logging methods are indicated, from which any two columns can be selected.

Module parameters



Logging data (LAS) file – Name of the file with logging curves. To select a file, click the **Browse...** button.

LAS column name(s) – the list of selected columns. Here, you can access the list of columns available in the file by clicking the **Edit** button and, in the following dialog window move the name of maximum two columns from the **Available** window to the **Added** window.



Load model file - a file with initial velocity model. A new one will be constructed on the basis of this model. The file format is the same as that of the file required for the **VSP Data Modeling** module, however, only the Z column is needed. By clicking the **Browse...** button the user can select a file in a standard dialog.

Save model file – the file to which the obtained finite velocity model will be recorded. The format is similar to that in the previous case. The Z, DEPTH, Vlay, Vmean columns contain receiver depth values starting with what is assumed to be zero cable depth, layer velocity and mean velocity, respectively.

Start Z (m), End Z (m) - define depth intervals (height scale) to be visualized in the window (all data can be viewed by scrolling).

Altitude correction (m) wellhead altitude.

Start time (ms), End time (ms) - define time interval for VSP data (horizontal scale) to be

visualized in the window (all data can be viewed by scrolling).

Trace scale - amplitude multiplier used during visualization.

Trace step (m) - depth step with which the output traces are to be selected.

Interval velocity calc. base - base (in traces) for interval velocities calculation.

Regularity parameter - visualization parameter used during velocity module calculation by layer stripping method. The increasing of this parameter results in obtaining smoother velocity values but less fitting to initial data on transmitted waves' time of arrival.

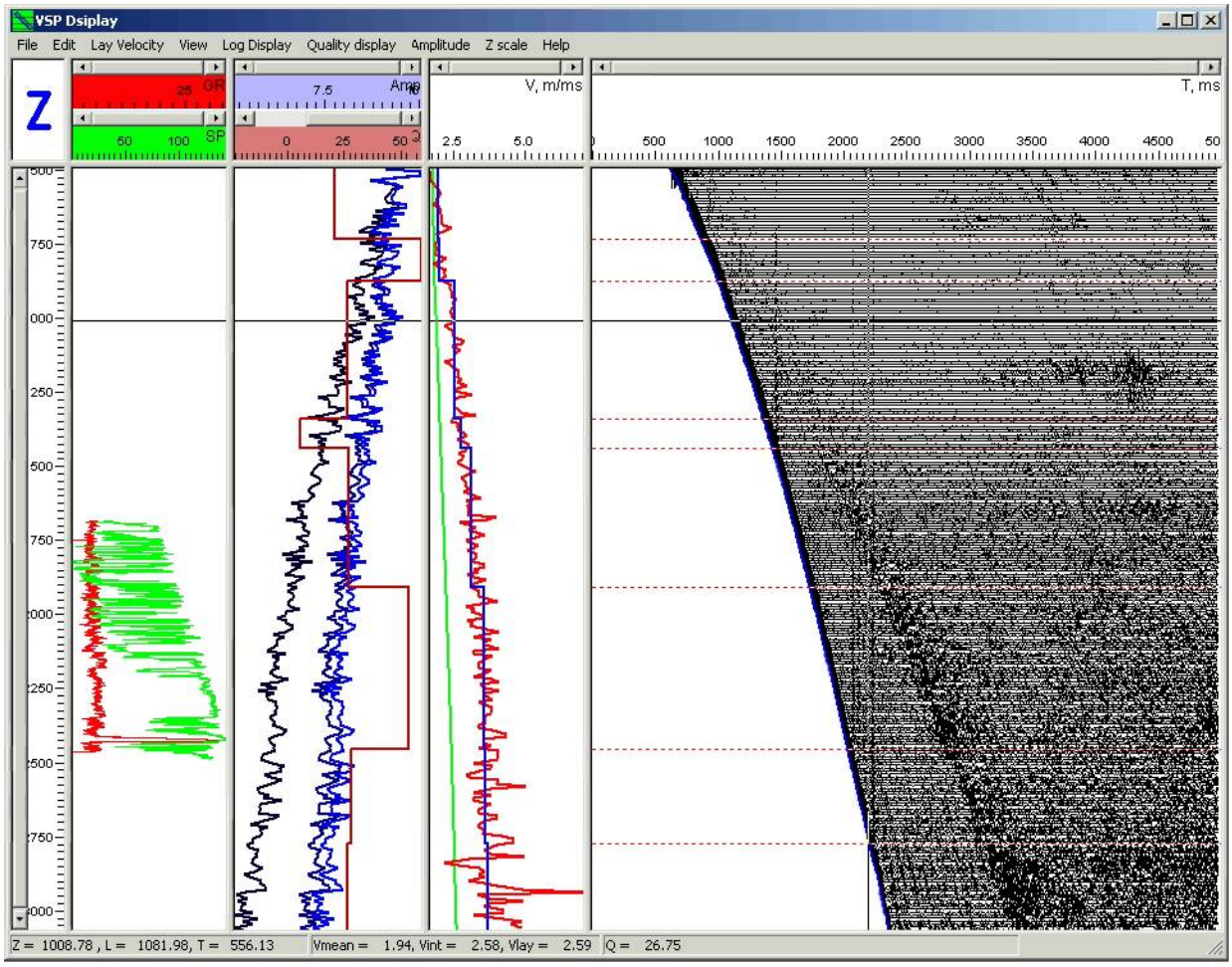
Attenuation/Get amplitudes from - traces header field from which the incident wave amplitude will be obtained during absorption calculation.

Working with the module

Main module window

When starting execution of the flow containing *Advanced VSP Display* module the window

similar to one shown on the figure will appear in the case where parameters have been specified correctly:



The window is divided into 5 parts and contains the following sections (from left to right):

depth (cable, if L is indicated in the header or true, if Z is indicated in the header); scale;

the set of Logging curves selected by the user (this column may be lacking);

the section with charts of measured amplitudes **(Amp)** and calculated mechanical quality (Q);

the column with velocity (V) charts;

and the main window with VSP seismogram put on vertical from reflected waves arrival-time curves and duplicated reduced arrival-time curve.

The logging curves are displayed by the same color as their names are displayed in the upper part of the column ***(red*** and ***green).***

Amplitude curves and Q values are displayed by:

black color - initial values of amplitudes;

dark-blue color - spherical divergence corrected amplitude values;

light-blue color - divergence and boundary passage corrected amplitude values;

vinous color - mechanical (acoustic) quality values **Q.**

Velocity curves are displayed by:

green color - mean velocity

red color - interval velocities calculated on continuous base

blue color - layer velocities

In the window with VSP seismogram the reduced arrival-time curve is displayed in ***dark-blue.***

In the status bar the values of two-way traveltime and depths for current cursor position are shown. In addition, for the layer, where the cursor is placed, elastic wave velocity and quality values are displayed.

Menu

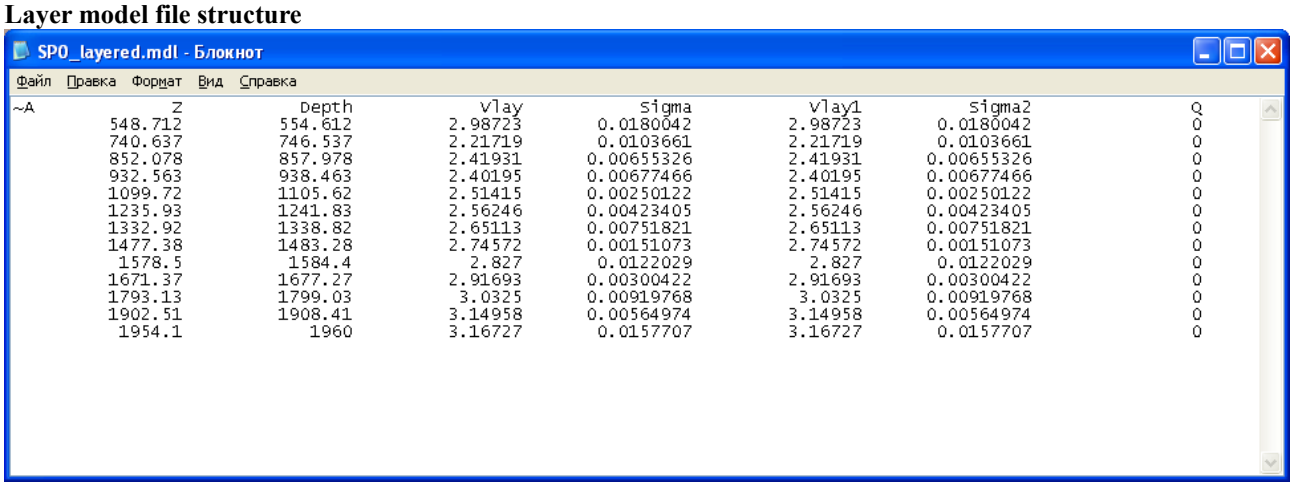
File contains two submenus:

Export results - allows the export of layer velocity model **(Layer model file)** and the per trace velocity model (**Per-trace file**) into ASCII files. The formats of the files is described in the NOTE box below.

Exit - allows exiting the module.

NOTE:

Layer model file structure



~A	Z	Depth	Vlay	Sigma	Vlay1	Sigma2	Q
	548.712	554.612	2.98723	0.0180042	2.98723	0.0180042	0
	740.637	746.537	2.21719	0.0103661	2.21719	0.0103661	0
	852.078	857.978	2.41931	0.00655326	2.41931	0.00655326	0
	932.563	938.463	2.40195	0.00677466	2.40195	0.00677466	0
	1099.72	1105.62	2.51415	0.00250122	2.51415	0.00250122	0
	1235.93	1241.83	2.56246	0.00423405	2.56246	0.00423405	0
	1332.92	1338.82	2.65113	0.00751821	2.65113	0.00751821	0
	1477.38	1483.28	2.74572	0.00151073	2.74572	0.00151073	0
	1578.5	1584.4	2.827	0.0122029	2.827	0.0122029	0
	1671.37	1677.27	2.91693	0.00300422	2.91693	0.00300422	0
	1793.13	1799.03	3.0325	0.00919768	3.0325	0.00919768	0
	1902.51	1908.41	3.14958	0.00564974	3.14958	0.00564974	0
	1954.1	1960	3.16727	0.0157707	3.16727	0.0157707	0

Layer velocity model file is an ASCII file of the following format:

First line starts with *~A*. Then this line defines the column names. The following columns can be present:

Z – vertical depth of the layer bottom in meters; *Depth* – cable depth of the layer bottom in meters, *Vlay* – velocity within the layer, estimated as the slope of a straight line fitting a set of first arrival time-curve values within the layer, *Sigma* – r.m.s. deviation of the time-curve values from the straigt line; *Q* – the *Q*-factor. If the well is vertical, the columns *Vlay1* an *Sigma2* replicate the *Vlay* and *Sigma*, respectively. Otherwise, *Vlay1*contains the velocities estimated by layer stripping, while *Sigma2* contains r.m.s. deviation of the time-curve values from a straigt line corresponding to *Vlay1* velocity.

Per-trace model file structure

SP0_pertrace.mdl - Блокнот

Файл Правка Формат Вид Справка

~A	Z	Depth	FBPICK	T	T1	Vint	Vmean
	464.1	470	173	326.995	0	0	2.76867
	474.1	480	175.5	332.481	0	0	2.78314
	484.1	490	178.5	338.894	0	3.16898	2.78949
	494.1	500	182	346.241	0	3.03963	2.78806
	504.1	510	184.5	351.671	0	3.04644	2.80188
	514.1	520	188	358.991	0	3.00868	2.80046
	524.1	530	191	365.344	0	2.7744	2.8065
	534.1	540	194.5	372.642	0	2.63775	2.80521
	544.1	550	198.5	380.889	0	2.42565	2.79698
	554.1	560	202.5	389.13	0	2.2938	2.78915
	564.1	570	207	398.326	0	2.17493	2.77496
	574.1	580	211.5	407.519	0	2.10956	2.76144
	584.1	590	216.5	417.673	0	2.06809	2.74219
	594.1	600	221	426.86	0	2.11095	2.73003
	604.1	610	226	437.01	0	2.1783	2.71239
	614.1	620	230	445.223	0	2.20234	2.70728
	624.1	630	234.5	454.399	0	2.2269	2.69662
	634.1	640	239	463.571	0	2.22799	2.6864
	644.1	650	243.5	472.741	0	2.20495	2.6766
	654.1	660	247.5	480.935	0	2.27918	2.67259

Per-trace velocity model file is an ASCII file of the following structure:

First line starts with *~A*. Then this line defines the column names. The following columns can be present:

Z – vertical depth of the current trace in meters; *Depth* – cable depth of of the current trace in meters; *FBPICK* - first break arrival time (ms); *T* – two-way vertical travel time calculated from the first break (ms), *T1* - two-way vertical travel time calculated from the velocity model (ms); *Vint* – interval velocity (km/s), *Vmean* – mean velocity (km/s).

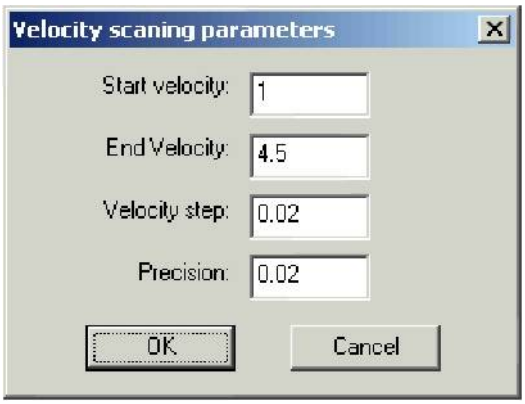
Edit contains one submenu: per-trace model **(Per-trace file):**

Parameters - allows setting up module parameters. The window for parameters setting - **VSP Display Parameters -** will open (see **Module parameters** chapter).

Layer Velocity - the menu providing for layer velocity calculation management by means of layered stripping. It contains two submenus:

Recalculate - when selecting this command the layer model is recalculated taking the refraction into account;

Scanning parameters - this command opens the window for velocity scanning parameters setting:



Here:

Start velocity, End velocity - start and end velocities in the model (expressed in km/s)

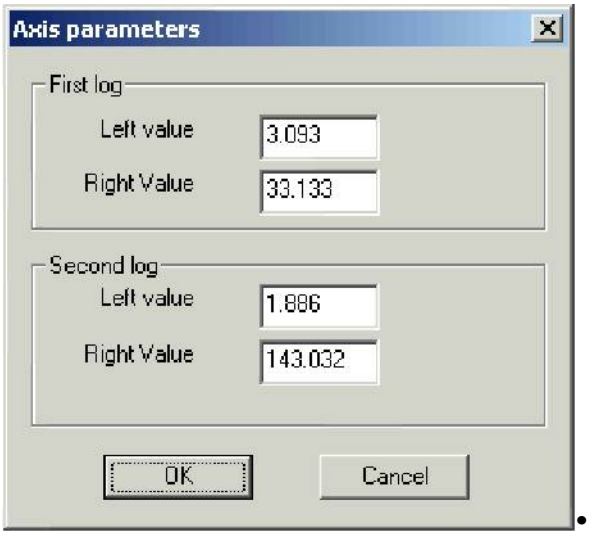
Velocity step - velocity selection step (expressed in km/s)

Precision - layer tracing precision

View menu allows chanting the displayed curves list.

Log display menu contains one submenu:

Axis **parameters** - when selecting this command the window for adjusting the dynamic range of axes displayed on logging curves, will open



Quality display - the menu for Q -factor calculation contains one submenu:

Frequency - selection of this command allows the user to specify the central frequency used for estimation of Q (Hz) calculation

Amplitude - the menu for amplitude displaying parameters changing contains the following submenus

Scale type - enables scale type selection between logarithmic type and linear one

Show - enables displayed curves selection: **Initial amplitude** - the initial values, with **SDC correction** - with spherical divergence correction, **True amplitude -** with divergence and boundary passing correction

Z scale - the menu allows depth scale selection (Z - vertical, **Depth** - cable).

Working with scales

Depth, time, parameter value axes are the elements of management that allow changing the

corresponding scale. To do this, click the left mouse button (**MB1**) on the start axis value and, holding it down, move the mouse to the position of the end axis value and release the button. To return to the initial scale on the selected axis, click on the corresponding axis with the right mouse button (**MB2**).

Layer boundaries editing

Layer boundaries can be added, deleted or displaced.

In order to add the layer boundary, click by the left mouse button (**MB1**) on the seismogram, where the boundary to be added.

In order to move the layer boundary, click the left mouse button (**MB1**) to the new position and then release the button.

In order to delete the boundary, double-click by the right mouse button (**MB2**) on the boundary.

Q-factor evaluation technology

While seismic wave propagation a part of energy is spent on inelastic deformations of medium -is absorbed. It is considered that wave amplitude decreasing caused by absorption can be described by the formula (Aki, Richards, 1983).

$$A(x) = A_0 \times \exp\left[-\frac{\omega x}{2cQ}\right], (1)$$

where x is wave travel path,

c is phase velocity,

w – circular frequency,

A – amplitude,

Q – quality factor determining absorption.

It should be noted that in our case a *spatial* parameter Q is used whereas in most of similar tasks the *time* parameter is used.

In order to simplify the task it is assumed that the Q parameter does not depend on frequency. As a rule, within seismic range of frequencies such assumption is quite acceptable and does not result in considerable errors.

Highly absorbent mediums, evidently, correspond to low Q values. In most of the cases the Q value is between 50 and 300 (Hatton, and others, 1989).

There are two absolutely different Q parameter evaluation techniques. The first one involves investigation of amplitude decreasing degree throughout the borehole taking divergence and transmission into account. The second one involves analysis of frequency spectra of incident waves.

In the module the Q parameter evaluation is accomplished from reflection amplitude variation.

From (1) it follows that $\ln(A(x)) = -\frac{\omega x}{2cQ} \times \ln(A_0)$. It means that if the signal is harmonic with

some frequency *w*, then, when dividing the curve of logarithmic amplitude of a signal on approximately linear sections and approximating them by line segments (via the smallest squares

method - the same as for layer velocities) and knowing the velocities, you could obtain the Q values. Since the real seismic signal, as a rule, is quasi-harmonic with pronounced central frequency, the rough estimation of the intrinsic attenuation can be obtained for it via the same method by substituting *w* by the central frequency of a signal.

This is how the acoustic quality is calculated in the module. During calculation, the amplitude values corrected for spherical divergence and transmission are used. The central frequency used while Q calculation is specified by the user in explicit form through the **Quality display/Frequency** menu.

To increase calculated Q value reliability, it is recommended to apply surface-consistent source and surface-consistent receiver corrections to amplitudes.

Surface-consistent source correction can be carried out using reference seismometer data. If the wavelet does not change much from excitation to excitation, then you can read the amplitudes of, for example, some pronounced phase of the direct wave, normalize them, and divide the VSP seismogram traces by the respective numbers. Otherwise, if the wavelet changes significantly, the surface-consistent source correction can be performed by means of deconvolution).

Surface-consistent receiver correction must at least take into account recording equipment gain coefficient variations during the acquisition. Besides that, while working with a multimodule instrument you must take into account that different modules in the instrument often differ distinctly in sensitivity. It is convenient to do corresponding corrections during the general preprocessing in a data processing flow.

VSP NMO

This module allows application of normal moveout corrections to VSP data obtained for horizontal layered model in deviated wells with offset source position.

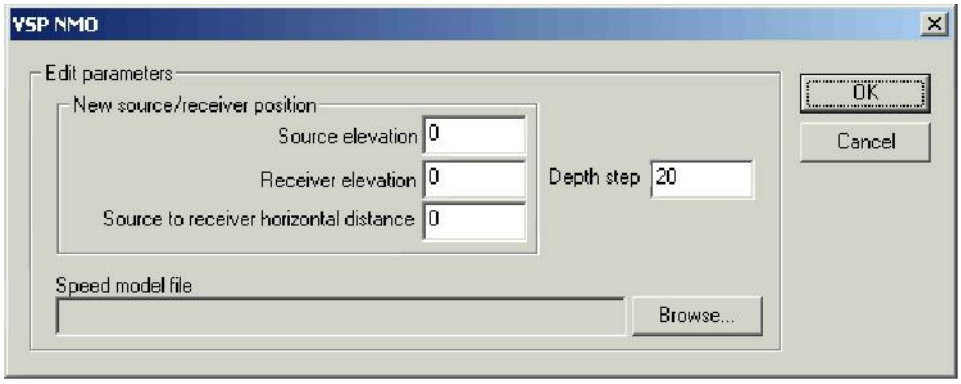
Theory

Obviously, while observations the VSP position of elastic waves source and receiver almost always do not coincide. When CDP processing by seismic reflection method you always trie to obtain the

section, which would be registered while source-receiver pair moving along the surface corresponding to the reference level. To do this at a certain stage, apply Normal Moveout Correction routine. Due to the fact that VSP observations are often applied for CDP data depth tie, it would be useful to transform VSP data before tie what will result in that every trace of vertical profile registered at a certain source-receiver position will obtain a form which it could have had if the source and the receiver were united and were at a certain level. This module transforms every trace to the form which is would have had if the source and receiver have been at position defined by the used (source elevation, receiver elevation, source to receiver horizontal distance).

The input data for the module is: VSP seismogram with given geometry (SOU_X, SOU_Y, SOU_ELEV, REC_X, REC_Y, REC_ELEV fields) and one-dimensional lay velocity model. As a result of transforms applied to the seismogram traces individually you obtain the seismogram corrected for normal moveout.

Module parameters



- **Source elevation** - new source elevation expressed in meters (0 is set by default).

- **Receiver elevation** - new receiver elevation expressed in meters (0 is set by default).

- **Source to receiver horizontal distance** - new source to receiver horizontal distance expressed in meters (0 is set by default).

- **Depth step** - algorithm depth step expressed in meters (20 m is set by default).

- **Speed model file** - velocity model file.

Speed model file - is an ordinary text file having the following structure: the data on velocity model

starting with the line specifying the names for the columns in the following way: *~A ... Z ... Vlay ...* or *~A ... Vlay ... Z.…* The *~A* field defines the start of velocity model setting in the file. Instead of dots you can enter other names of other columns. In the next lines there should be values of respective columns organized in the order they are specified in the header.

Algorithm

For corrections calculation and application, the following transform should be applied. The model is treated as horizontally-layered. This means that the medium is assumed as a half-space restricted by horizontal plane (the half-space is below the boundary) and divided on layers with boundaries parallel to half-space boundary. Elastic wave propagation velocity (only one velocity is important in this task) inside the layers is constant.

Let's assume that the source is in the point with SOU_X, SOU_Y, SOU_ELEV coordinates and the receiver is in the point with REC_X, REC_Y, REC_ELEV coordinates. In this case the obtained trace must correspond to the source and receiver position: SOU_ELEV_new, REC_ELEV_new, SOU_ELEV_dist (source to receiver horizontal distance).

Having started with the greatest depth of the source and receiver we will move downward with a specified step. For every obtained trace we calculate the time require to the ray to cover the distance between the source and receiver after reflecting from the boundary situated on this depth. In case if the ray can not cover the distance from source to receiver after reflecting from the desired depth (total internal reflection) the corresponding amplitude on the trace will be muted.

This algorithm is applied to old and new position of source and receiver. As a result we obtain two time sequences corresponding to certain depths. After that, by means of interpolation we can obtain the trace corresponding to CDP trace.

QC (quality control)

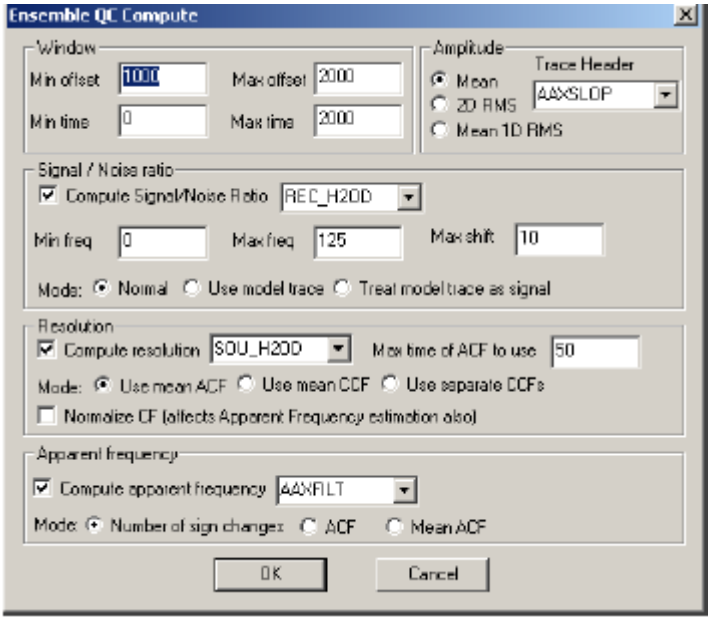
Ensemble QC

Ensemble QC Compute is designed for the computation of the main parameters that allow estimating the quality of seismic data in a certain time window and limited by certain offset range.

A traces ensemble is input to the module, the header field OFFSET contains the offset value. The output data represent parameters of seismic record, which are saved in header fields defined by user. If there are some reasons of failure to determine parameters, the header fields will contain value -1.0. The module fills the indicated header fields of all ensemble traces, not only those that fall within the given offset range.

The module allows calculating an average absolute (or mean square) amplitude in a window, signal/noise ratio, estimation of the record resolution and apparent signal frequency.

Dialog of module options is represented on the figure:



Module parameters (and, correspondingly, modes of the module):

- **Window group**

Min/Max offset . minimum and maximum offset

Min/Max time . minimum and maximum time

The module picks the data that fall within the window from the data that are being processed in the flow, that's why there is no necessity to limit the offset range while sorting (it is well understood that it should not be narrower than the estimation range).

- **Amplitude group**

Trace Header . trace header name to save the window amplitude estimation

Mean . mean absolute amplitude will be calculated $A_{MEAN} = \frac{1}{NT}\sum_{i=1}^{N}\sum_{j=1}^{T}|Aij|$ (**Ai,j** . j[th]. – sample of i[th] trace, **N** . trace number in window, **T** . samples number in window)

1D RMS . the root mean square amplitude will be calculated for each trace, simple average of that kind of trace estimation will be used for the evaluation.

$$A_{1DRMS} = \frac{1}{N}\sum_{i=1}^{N}\sqrt{\frac{1}{T}\sum_{j=1}^{T}A_{ij}^{2}}$$

2D RMS . root mean square amplitude in window

$$A_{2DRMS} = \sqrt{NT}\sum_{i=1}^{N}\sum_{j=1}^{T}A_{ij}^{2}$$

Signal / Noise ratio group. The estimation of signal/noise

Compute signal/noise ratio . allows to switch on the calculation of signal to noise ratio and to select trace header field to output the value.

Min/Max freq . minimum and maximum frequency in the range, which is used for signal-to-noise ratio calculation

Max shift . maximum shift between the traces where we find the maximum of cross-correlation function

Mode . mode of usage

Normal . cross-correlation functions are calculated inside the ensemble between the neighboring traces as well as autocorrelation for each trace, mean cross-correlation function and autocorrelation functions are calculated. It is assumed that averaged cross-correlation function autocorrelation function of a signal, while autocorrelation is autocorrelation of signal + noise. Then module

calculates a spectrum of averaged cross correlation function $S_{CCF}(f)$ and autocorrelation SACF(f), a signal to noise ratio is determined from the formula:

$$S/N = \sum_{f=f\,\min}^{f\,\max} \frac{S_{CCF}(f)}{S_{ACF}(f) - S_{CCF}(f)}$$

- **Use model trace** differs from the **Normal** mode, by the methods of calculation: the pairwise correlation is calculated between ensemble traces and a model trace, obtained by averaging of all ensemble traces.

- **Treat model trace as signal**. In this mode we treat model trace as signal free of noise. Its autocorrelation function is considered to the autocorrelation function of a signal.

- **Resolution** . estimation of data resolution power

The estimation is carried out using the formula: $\lambda = \frac{1}{2q} \times \frac{\sum_{i=1}^{T} A_i^2}{\sum_{i=q}^{N} A_i^2}$

where 2q - is the width of the principal half-period of the autocorrelation or cross correlation functions, Ai samples of autocorrelation function.

- **Compute resolution** allows calculating the estimation of resolution power and indicate the trace header field to which the value will be written (the value will be in Hz)

- **Max time of ACF to use** – duration of autocorrelation function, that can be used for resolution estimation

- **Mode**

- **Use mean ACF** . uses averaged autocorrelation function

- **Use mean CCF** . uses averaged cross correlation function

Use separate CCFs . uses cross correlation function of separate traces, the obtained estimation have to be averaged

Normalize CF . normalizes correlation function using averaging (this parameters impact as well on the apparent frequency estimation, if the it is calculated from the correlation distance)

Apparent Frequency . apparent frequency estimation

Number of sign changes . apparent frequency estimation on the basis of zero crossing number. Calculated for each trace using the formula, the values are averaged within the window.

$$f_{ZC} = \frac{N_{ZC} - 1}{2(t_{last} - t_{first})}$$

ACF . apparent frequency estimation on the basis of ACF main half period width. Calculated for each trace using the formula, $f_{ACF} = \frac{1}{2q}$ the values are averaged within the window

Mean ACF . the same as the previous mode, however in this mode the program uses averaged ACF. If the Normalize CF mode is on, then the ACF are normalized.

References

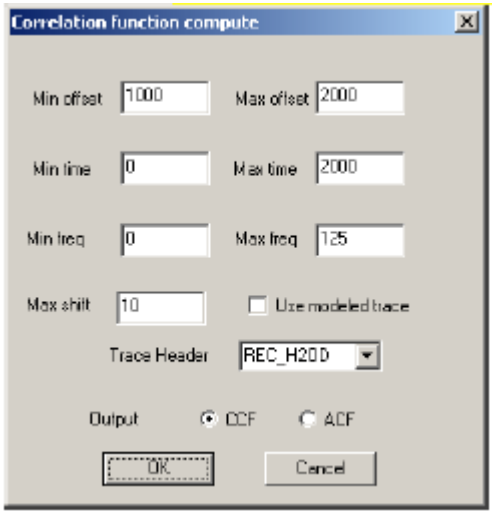
1. Katz, Ptetsov S.N. Spectral analysis of regular signal and noise field// Physika Zemli, 1978, №1 (in Russian)

2. Hardy H.H., Beier R.A., Gaston D.J., 2003, Frequency estimates of seismic traces //Geophysics, 68, no. 1, pp. 370-380

Correlation function compute

The module is designed for the computation of various correlation functions estimation, for the indicated offset and time ranges. The trace ensembles are input to the module with the header field OFFSET containing the offset value. As a result, each ensemble is replaced by an averaged ACF or CCF. Moreover, the output traces headers contain the signal to noise ratio estimation.

The parameters options dialog look like:



Parameters:

- **Min/Max offset** . minimum and maximum offset

- **Min/Max time** . minimum and maximum time

The module picks the data falling within the window from the data that are processed in the flow, that why there is no need to limit the offset range while sorting (it is well understood that it should not be narrower than the estimation range).

- **Min/Max freq** . minimum and maximum frequency (Hz) in the range, used for signal to noise ratio estimation

- **Max shift .** maximum shift between the traces where do we estimate the maximum of cross correlation function

- **Use modeled trace** . if this mode is toggled off, the cross correlation function will be calculated between the pairs of the neighboring traces, if it is toggled on . between stacked trace and each ensemble trace.

- **Trace Header** . header field, containing the calculated signal/noise ratio (the details are given in Ensemble QC Compute).

- **Output** . this parameter toggles between the output type.

  - **CCF** . the output of cross correlation function,

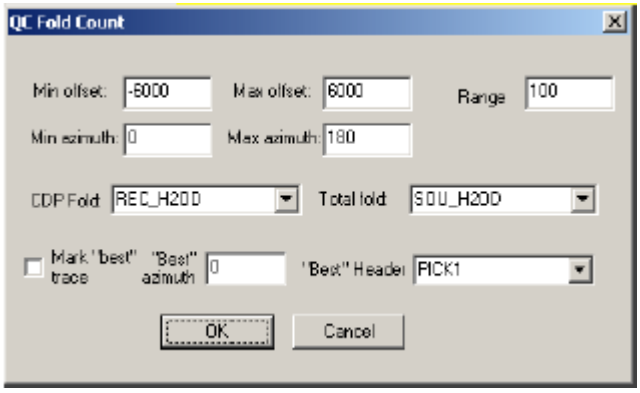
  - **ACF** . autocorrelation function.

Compute fold

The module is designed for the calculation of fold coverage. The trace ensembles are input to the module, the OFFSET trace header field contains the offset value, the header fields REC_X and REC_Y . contain receiver coordinates, SOU_X, SOU_Y - source coordinates. The values of general and effective fold count in the given azimuth range acquired as a result are output to header fields indicated by user.

General fold count is overall trace number in the ensemble (bin), the effective - is the number of nonempty offset -bins in the ensemble (bin). Traces inside the ensemble are distributed by groups in accordance with offset for the calculation of the effective fold count. Offset range falling within one group is set by the **Range** parameter, the groups do not overlap, there is no gap between the groups. Effective fold count is the number of nonempty groups.

The figure represents the module options dialog.



Module's parameters:

- **Min/Max offset** . minimum and maximum offsets

- **Range** . offset range, falling into one offset-bin

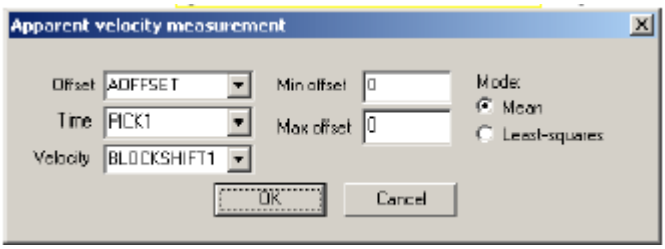
- **Min/Max azimuth** . minimum and maximum azimuths (in degrees, Y-axis is directed to the North)

- **CDP Fold** . header field of effective fold count

- **Total fold .** header field of general fold cover

Apparent velocity calculation

This module is designed for calculation of apparent velocity of direct wave from the arrival times inside the trace ensembles. There are two methods of calculation : mean velocity calculation and approximation of direct waves arrival using the least squares method. The resultant value, obtained from the calculation is output to the header field of ALL ensemble traces indicated by the user



Module parameters:

- **Offset** . header field, containing source to receiver distance

- **Time** . header field, containing arrival times

- **Velocity** . header field, containing velocity value

- **Min/Max offset** . minimum and maximum offsets (the same header field indicated in Offset parameter is used), that are allowed to use for velocity estimation.

- **Mode.**

In **Mean** mode velocity computation is carried out using the formula $velocity = \frac{1}{N}\sum_{i=1}^{N}\frac{offset_i}{Time_i}$

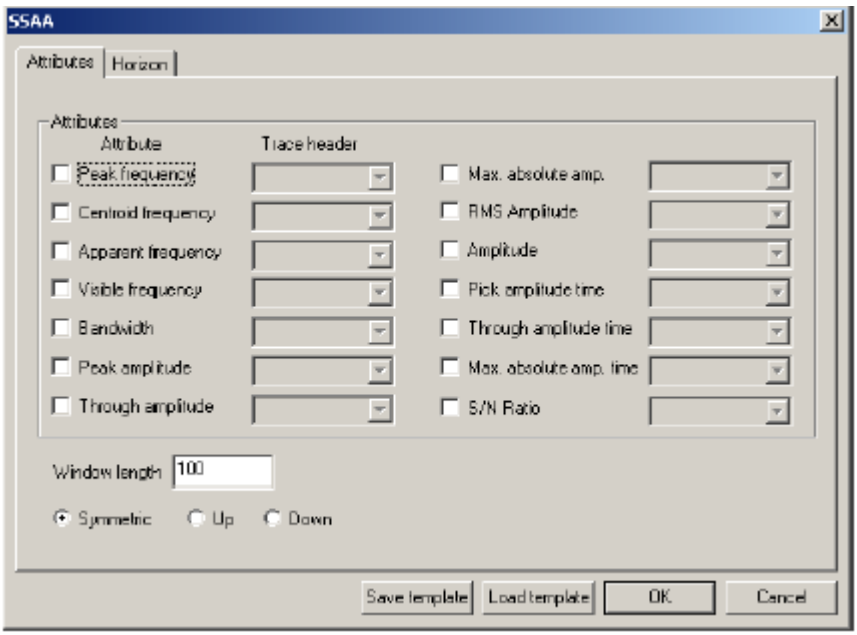
In **Least-squares** . using least squares method from the expression

$$velocity = \frac{\sum_{i=1}^{N} Offset_i \times Time_i - \frac{1}{N}\left(\sum_{i=1}^{N} Offset_i\right)\left(\sum_{i=1}^{N} Time_i\right)}{\sum_{i=1}^{N} (Time_i)^2 - \frac{1}{N}\left(\sum_{i=1}^{N} Time_i\right)^2}$$

SSAA (Seismic Sequence Attribute Analysis)

The module calculates the traces attributes in window along the indicated horizon. The calculated seismic attributes are written to database headers.

The following window emerges when you activate the module, it includes two tabs: Attributes and Horizon.



- **Attributes** field contains the following seismic attributes:

- **Peak frequency**. Frequency that corresponds to the principal maximum in spectrum.

- **Centroid frequency**. Mean frequency, calculated from the formula: $fc = \frac{\int_0^{f_N} f|S(f)|df}{\int_0^{f_N} |S(f)|df}$

(here S(f) – spectrum in time window, where the frequency estimation is carried out, f – frequency, fN – Nyquist frequency).

- **Apparent frequency**. Apparent frequency, acquired from the estimation of number of zero crossings: $f_{NZ} = \frac{N_{ZC} - 1}{2(t_{last} - t_{first})}$

(Here N_{ZC} – number of zero crossing, t_{last} – time mark of the last zero crossing, t_{first} – time mark of the first zero crossing)

- **Visible frequency**. Apparent signal frequency, estimated from correlation distance.

- **Bandwidth**. Frequency range.

- **Peak amplitude**.

- **Through amplitude**.

- **Max. absolute amplitude**. Maximum absolute amplitude value.

- **RMS amplitude**. Root mean square amplitude,

- **Amplitude**.

- **Peak amplitude time**.

- **Through amplitude time**.

- **Max. absolute amplitude time**. Time mark corresponding to the maximum absolute amplitude.

- **S/N ratio**. Signal to noise ratio, estimated from the normalized cross correlation between pair of neighboring traces.

Trace header field is activated when you select an attribute; in this field you have to indicate header to which the calculated attribute will be saved.

- You have to indicate the length of time window in ms to calculate seismic attributes in the field **Window length**. With respect to the given horizon the time window can be:

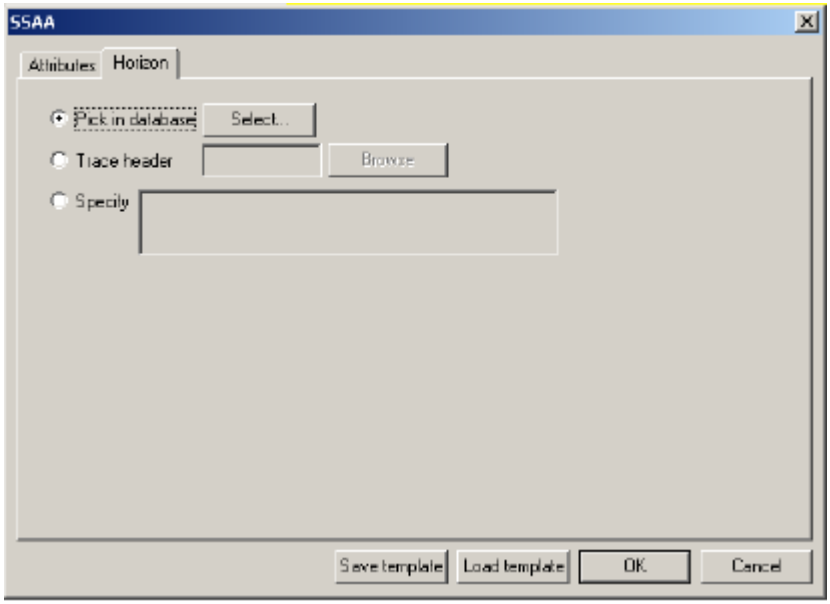
- Symmetric

- Up

- Down .

- Horizon parameters with respect to which the attributes calculation window will be positioned

are set in the tab Horizon.



The horizon is set by three methods:

- **Pick in database**. Specify the horizon from the database. To do so activate the option, click the button **Select…,** then choose the right file from the pop up window.

- **Trace header**. Specify the horizon from the header, where is has been saved to. Activate the option, click the button **Browse**, you have to select the needed header in the dialog window.

- **Specify**. Specify the horizon manually in the field Specify.

An example of horizon specification:

CDP

100 2250

In this example:

CDP – header field, by which the horizon is specified;

100 – header field value (in this case the number of CDP),

2250 – time mark of horizon in the given point.

Save template and **Load template** buttons are designed for saving current parameters of module in the template of project database and loading parameters from the previously saved template, correspondingly.

CrossPlot (Tied crossplots)

This module is used to analyze interrelations between header fields using crossplots and histograms.

A crossplot is built based on a pair of seismic trace header fields, with each point on the crossplot corresponding to one trace. A histogram shows distribution of a single seismic trace header across value ranges; the height of each column corresponding to a particular range is determined by the number of traces with the selected header field values falling within that range.

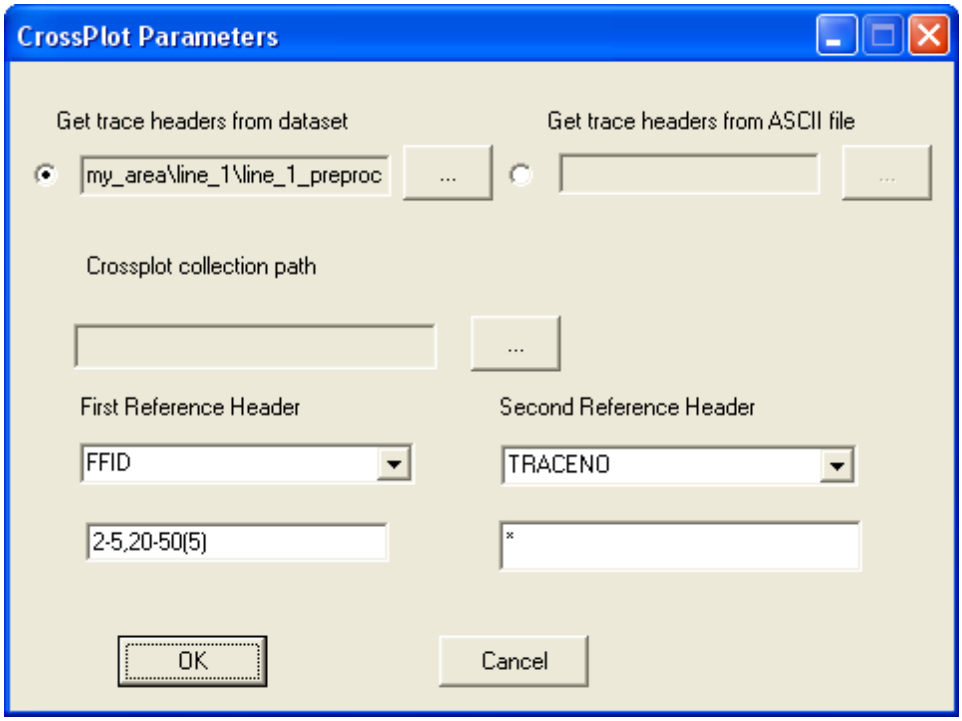
The combination of all crossplots/histograms tied to one dataset together with all their parameters is called a crossplot collection. The collection is stored in the project as a database object. Therefore, the module works with a crossplot collection.

The user can select a so-called active crossplot (or histogram) which allows interactive highlighting of various areas of interest with different colors. Highlighting an area on a crossplot is done by defining a polygon enclosing that area; on a histogram, it is done by specifying the corresponding interval. When the polygons are specified on the active crossplot, the points on all other crossplots assume the colors of polygons within which the traces corresponding to those points fall on the active crossplot. Similarly, when the intervals are specified on the active histogram, the points on all other crossplots within the collection assume the colors of intervals containing the corresponding traces.

If necessary, a topographic base can be used as the crossplot background. The module also allows printing arbitrary sets of crossplots and histograms with added titles and legends.

CrossPlot is a so-called standalone module, i.e. a module that generates the flow independently. Therefore, the module should be the only one in the flow, with no presence of additional modules required.

Module parameters



When the module is added into the flow, the **CrossPlot Parameters** dialog box appears:

The header fields that will be used to build the crossplots (histograms) may be selected both from the project database (the **Get trace headers from dataset** field, the **...** button) and from an ASCII file (the **Get trace headers from ASCII file** field, the **...** button).

Specify the name under which the crossplot collection will be saved to the database or select an existing name (the **Crossplot collection path** field). If you leave this field empty, a collection object will be created under the current flow name by default.

Instead of using all traces from the dataset to analyze interconnections between the header fields, you can choose to use only those that fall within a certain range. The range is defined by two reference header fields.

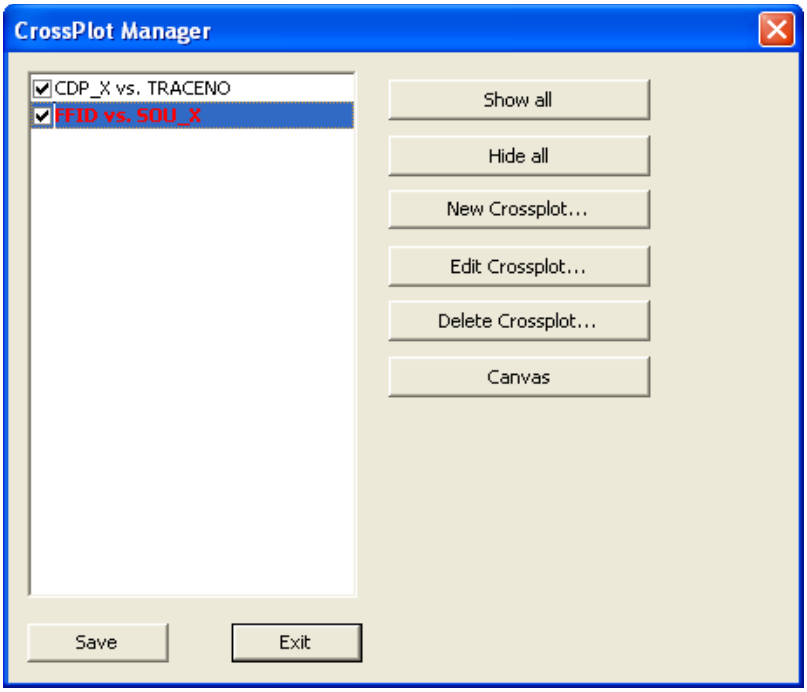
Select the reference header fields from the list (**First Reference Header, Second Reference Header**) and specify the corresponding value ranges for them (much in the same way as header field sorting is specified in the **Sort Fields** field of the **Trace Input** module). Defining the ranges by reference header fields allows analyzing the headers of only those traces that fall within the selected range instead of all traces in the dataset.

When you are done setting the module parameters, press **Ok**.

Working with the CrossPlot Manager

Press **Run** to start execution of the flow**.** The **CrossPlot Manager** window will open**.** A list of objects (crossplots and histograms) existing in the collection will be shown on the left side of the window. Visible objects, i.e. crossplots and histograms currently displayed on the screen, will be checked. You can toggle object visibility by left-clicking the corresponding checkboxes. The currently active crossplot/histogram will be highlighted in red.

The **Show all** button allows making all crossplots in the list visible. The **Hide all** button hides all crossplots.



You can also control crossplots and histograms with the following buttons available in the window:

- **New Crossplot... –** create a new crossplot/histogram.

- **Edit Crossplot... –** edit the parameters of the crossplot/histogram selected in the list.

- **Delete Crossplot...** – delete the selected crossplot/histogram from the list.

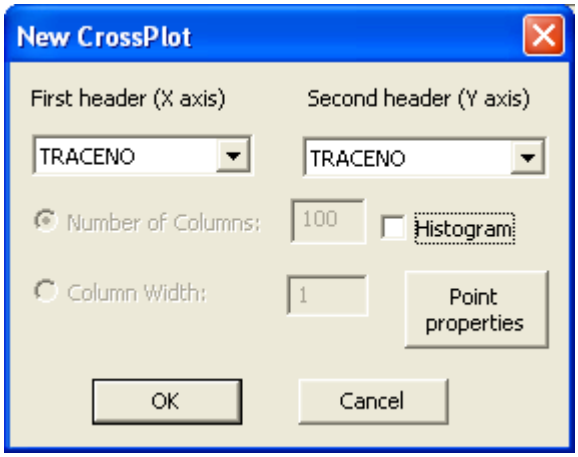
- **Canvas** – create a "canvas" with a set of crossplots and histograms as well as additional information for printing.

- **Save –** save the current crossplot and histogram set.

- **Exit** – exit from the module. When this button is pressed, the program will prompt you to save the crossplot and histogram set before exiting.

Creating a new crossplot/histogram

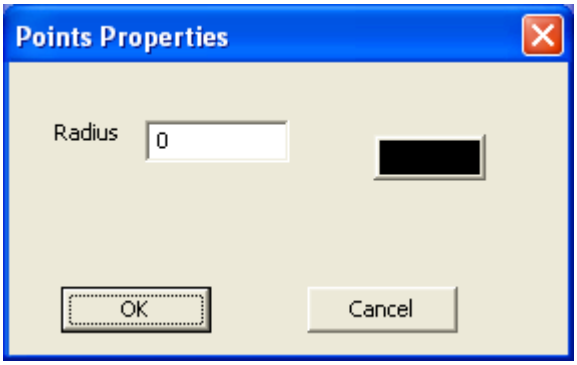
Press the New Crossplot... button in the CrossPlot Manager dialog box. The following new crossplot/histogram setup dialog box will appear:



Histogram – a histogram will be generated if this option is enabled; otherwise a crossplot will be generated.

First header (X axis), Second header (Y axis) – header fields that will be used to build the crossplot/histogram (if a histogram is to be generated, only one header field will be available).

Point properties / Histogram Color – display parameters for crossplot points/histogram columns. If a crossplot is to be generated, the point properties dialog box will look like this:



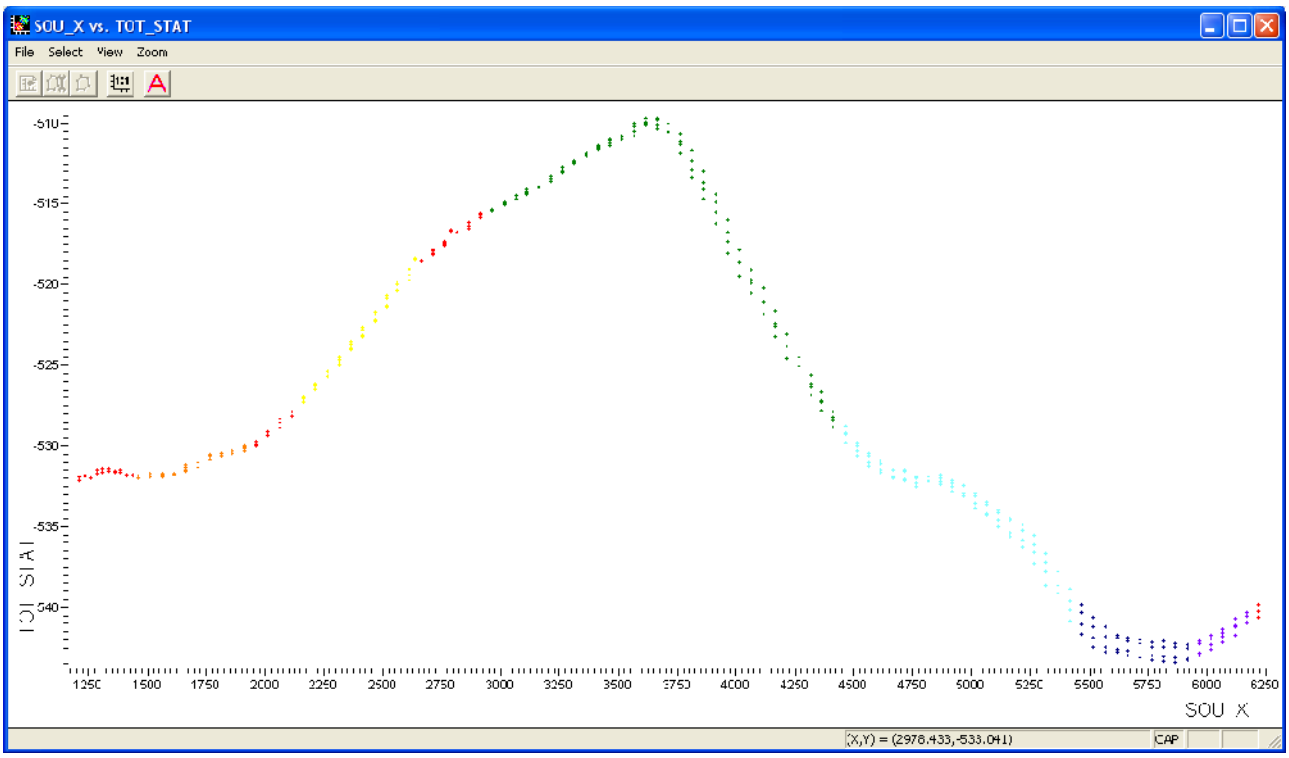
Radius – non-negative number defining the crossplot point display method. 0 – point size is 1 pixel, 1 – points are displayed as crosses, 2 and more – points are displayed as circles with the specified radius in pixels.

▪- the field is used to select the crossplot point color.

If a histogram is to be generated, only the histogram column display color selection field will be available.

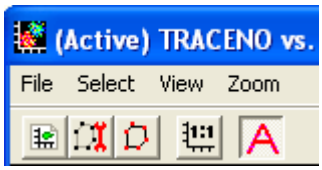
When a histogram is built, additional histogram parameters become available. You can specify either the number of columns (**Number of Columns**) into which the entire range of the selected header field values will be broken down, or the histogram column width (**Column Width**).

After a new object is created, its window immediately appears on the screen. An example of a crossplot window is shown below:



And this is an example of a histogram window:

Crossplot/histogram window



This is what the window's main menu and panel look like:

Here we will list all available commands and provide a short description for each one. The most important commands will be discussed in greater detail below.

- **File**. This menu contains the following commands:

- **Active plot –** select the current crossplot/histogram as active. The same is achieved by pressing the button on the toolbar (see the **Working with active crossplots/histograms** section).

- **Load Background Image** – load a bitmap background image.

- **Unload Background Image** – unload a loaded background image.

- (These commands are discussed in detail in the **Loading and unloading bitmap background images** section).

- **Export Image –** export the crossplot/histogram to a bitmap image.

- **Print...** – print the current crossplot/histogram.

- **Close** – close the current window and save its parameters.

- **Select**. Commands from this menu are available only if the current crossplot/histogram is selected as active. They allow working with crossplot polygons or histogram intervals. (These commands are discussed in detail in the **Working with active crossplots/histograms** section).

- **New Polygon/New Interval** – create a new crossplot polygon/histogram interval. The same is achieved by pressing the button on the toolbar.

- **Active Polygon's Properties/Active Interval's Properties** – set up the active polygon/interval properties. The same is achieved by pressing the button on the toolbar.

- **Edit Active Polygon/Edit Active Interval –** edit the active polygon/interval boundaries. The same is achieved by pressing the **Edit Active Polygon** button in the **Polygons Manager** window or the button on the toolbar.

- **Auto add intervals** – automatically break the histogram down into intervals – either by quantiles or uniformly.

NOTE

In mathematical statistics, a quantile is a number which may not be exceeded by a specified random

variable only with a fixed probability.

- **View** – this menu is used to set up the crossplot/histogram display window parameters.

Show Tool Bar – show/do not show the toolbar.

Show Status Bar – show/do not show the status bar.

Point Properties/Hystogram Color – open the **Point Properties /Histogram color** windows described in the **Creating a new crossplot/histogram** section.

Prompt Headers – this option allows displaying the values of additional header fields corresponding to the current crossplot point in the status bar. (This command is discussed in detail in the **Displaying the values of header fields corresponding to a crossplot point in the status bar** section).

- **Zoom** This menu contains the following options active only for the crossplot window:

Set axis ratio 1:1 – set the same scale on the X and Y axes (recommended to use for header fields of equal dimension, such as X and Y coordinates). The same is achieved by pressing the

button on the toolbar.

Keep ratio on square select – keep the aspect ratio when scaling the image up and down. (Image scaling is discussed in detail in the **Image scaling and working with the axes** section).

Editing the point size and polygon/histogram color

To edit the point size and crossplot or histogram display color, select the **View/Point Properties (View/Hystogram Color)** menu item. For a detailed description of the dialog box, see the **Creating a new crossplot/histogram** section above.

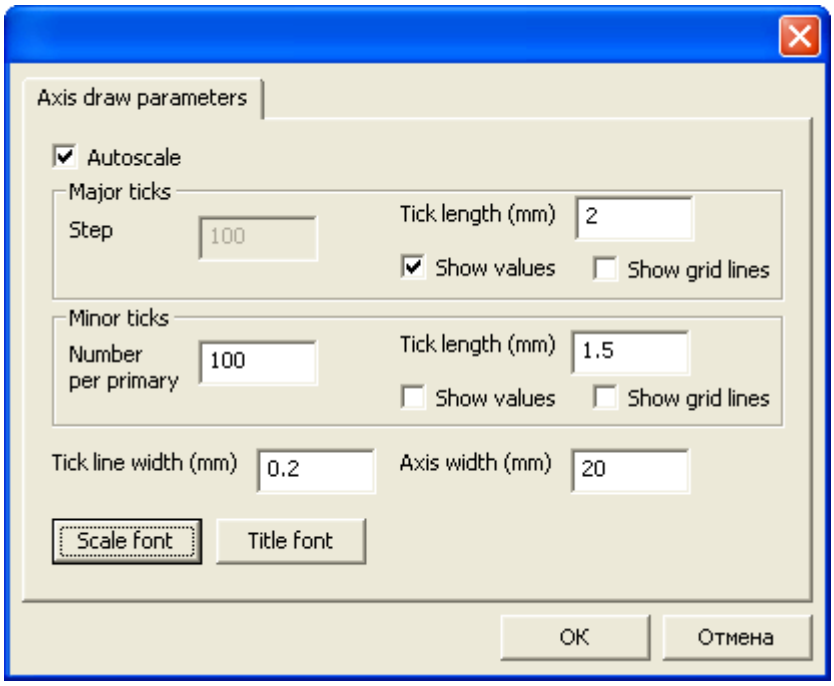
Image scaling and working with axes

To enlarge a fragment, press the left mouse button in the corner of the fragment you want to enlarge, drag the cursor to the opposite corner of the fragment while holding the mouse button down, and release the mouse button. The selected rectangular fragment will be scaled up to fill the entire window.

You can achieve the same result by selecting the relevant section on the horizontal or vertical ruler located at the top or left edge of the visualization window, respectively. To do this, press and hold down the left mouse button at the start of the section on the ruler, drag the cursor to the end of the section, and release the mouse button. The range selected on one of the rulers will be scaled up to full-screen size (the scale on the other ruler will remain unchanged).

To revert to the original scale (i.e. to make the entire image area fit into the view window), double-click the right mouse button in the crossplot/histogram window.

To set up the axis display parameters, double-click the left mouse button on the axis. The following dialog box will appear:



- **Autoscale** – when this option is enabled, the axis scale is determined automatically based on the axis length and the header field value variation range.

- **Major ticks –** major scale tick parameters.

- **Step –** tick interval in units corresponding to the header field. This field is available only if the **Autoscale** option is disabled.

- **Tick length (mm)** – tick length in mm.

- **Show values** – show/do not show scale values.

- **Show grid lines** – show/do not show grid lines.

- **Minor ticks** – minor scale tick parameters.

- **Number per primary** – number of minor scale ticks per primary tick.

- The **Tick length (mm), Show values** and **Show grid lines** parameters are similar to the ones described above.

- **Tick line width (mm)** – tick line width in mm.

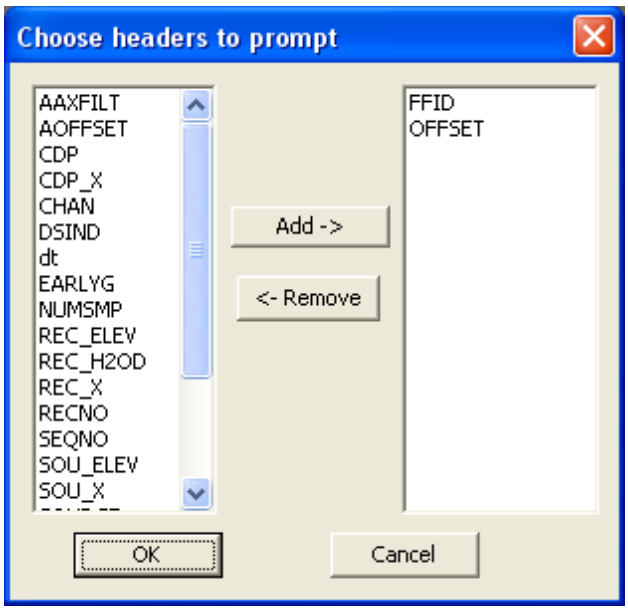
- **Axis width (mm)** – axis width in mm.

- **Scale font** – this button opens a standard dialog box allowing you to select the axis tick label font parameters.

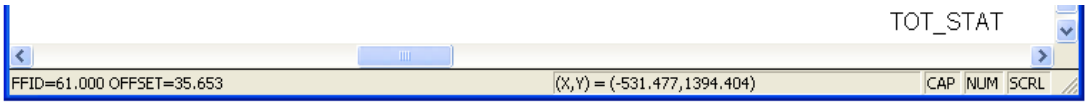
- **Title font** – this button opens a standard dialog box allowing you to select the axis title font parameters.

Displaying the values of header fields corresponding to a crossplot point in the status bar

This option is available through the **View/Prompt Headers** menu command in the crossplot display window. When it is selected, the **Choose headers to prompt** dialog box appears.



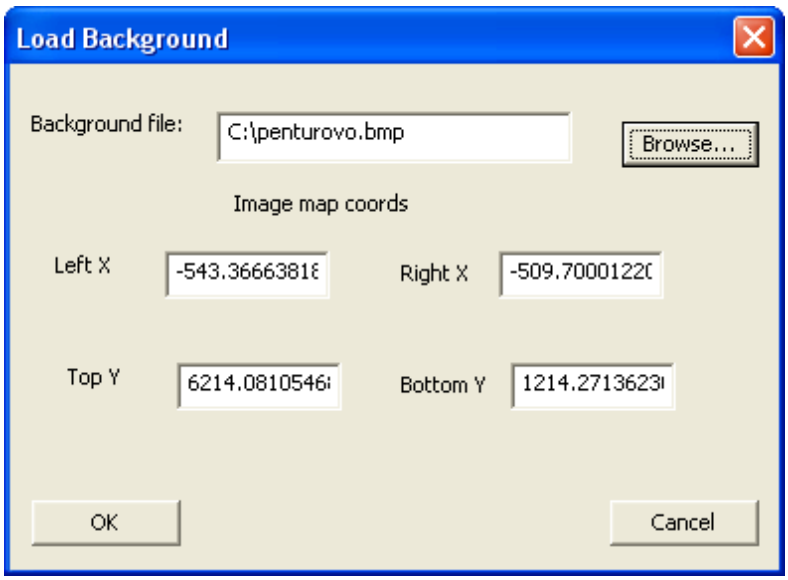
The left part of the dialog box contains the list of all headers in the dataset, the right part – the list of those headers whose values will be displayed in the status bar. Press **Add->** to add headers to the list on the right; press **<-Remove** to remove headers from that list. When you are done selecting the headers, press **Ok**. Now when you hover the mouse cursor over a point on the crossplot, you can see the values of the header fields from the list assigned to the currently selected point in the status bar. An example is shown below:



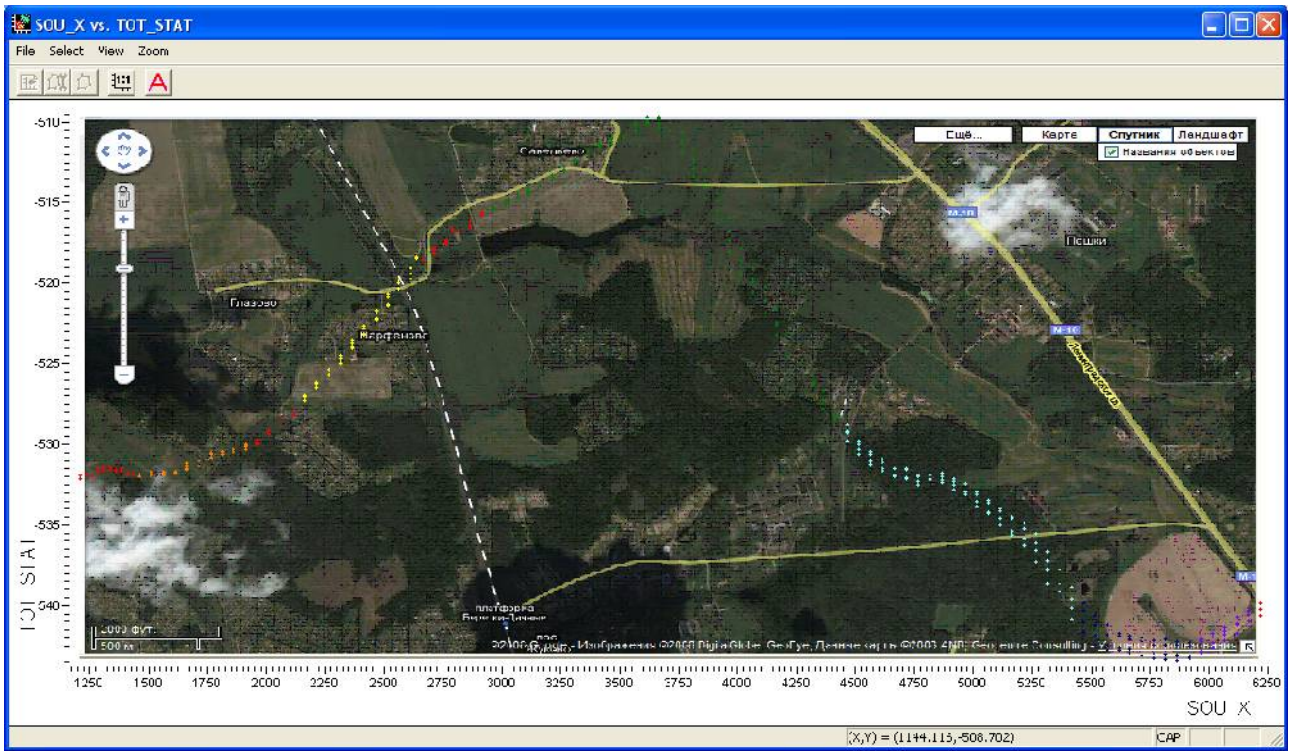
Loading and unloading bitmap background images (File/Load Background Image, File/Unload Background Image)

The **File/Load Background Image** and **File/Unload Background Image** menu commands allow loading or unloading bitmap images to be used as the crossplot background (for example, if coordinates are used as crossplot axes, a topographic map of the project area may serve as the background). The **Load Background Image** and **Unload Background Image** commands are available only when working with the crossplot window.

When the **File/Load Background Image** command is selected, the following dialog box appears:



Select the bitmap background image in the **Background file** field using the **Browse...** button. The following formats are supported: BMP, JPEG, GIF, TIFF, PNG. Set the edge coordinates of the image being loaded in the **Left X, Right X, Top Y** and **Bottom Y** fields: left and right edge along the X axis and upper and lower edge along the Y axis. By default these fields contain coordinates matching the minimum and maximum values on the X and Y axes on the crossplot. In the example shown below an aerial photograph is used as the background image.

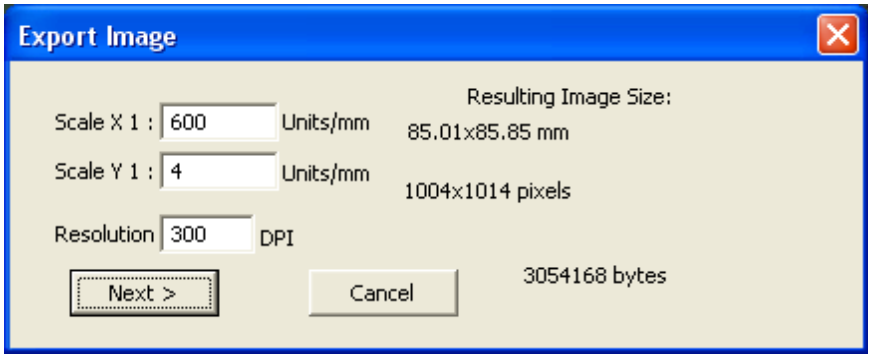


You can change the background corner coordinates at any time by re-opening the **Load Background** dialog box**.**

To unload an existing background image, select the **Unload Background Image** menu item**.**

Exporting crossplots/histograms to bitmap images

To export a crossplot/histogram to a bitmap image, select the **File/Export Image...** menu command. The following dialog box will appear:



Scale X 1 – horizontal scale, in units on the X axis per mm.

Scale Y 1 – vertical scale, in units on the Y axis per mm.

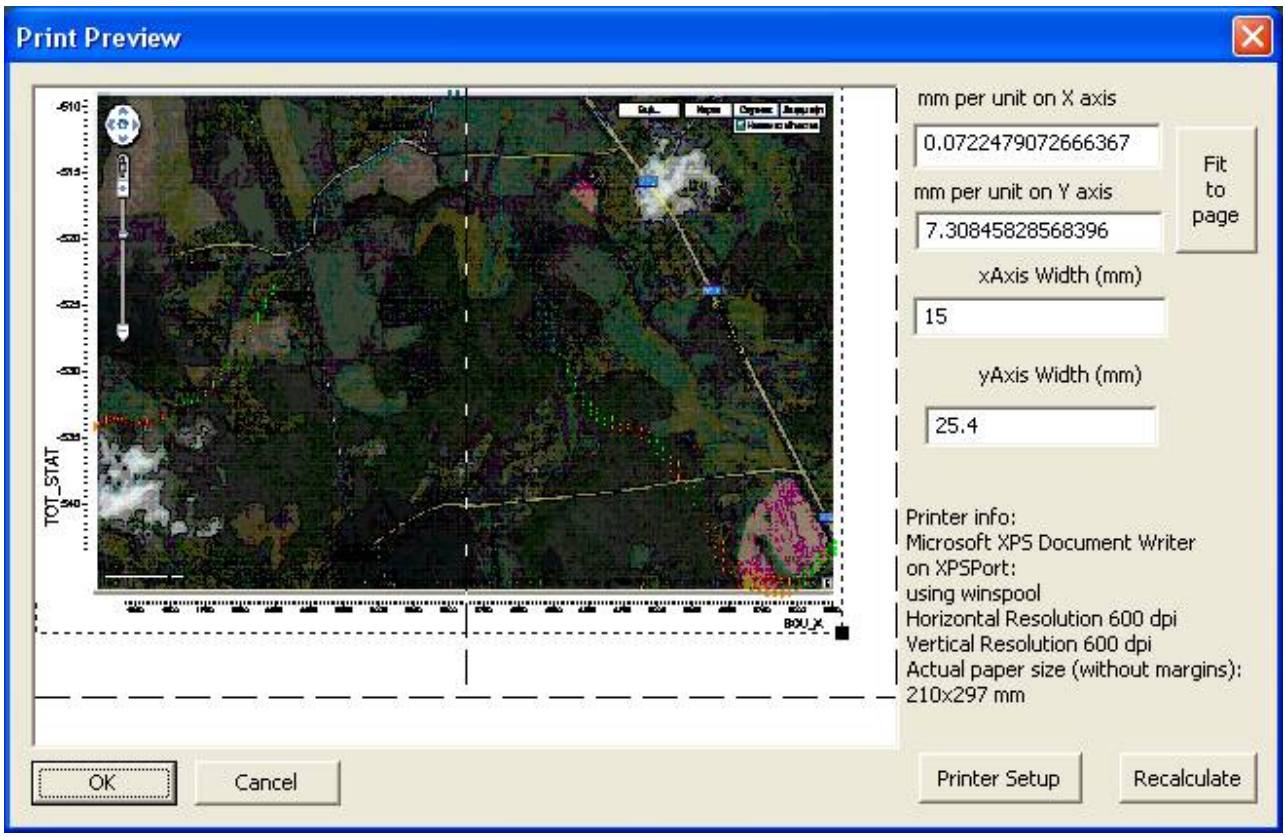
Resolution – image resolution in dots per inch.

Information on the image size in mm, resolution in pixels and file size in bytes (uncompressed) will be displayed on the right side of the dialog box.

When you are done setting the parameters, press **Next** and select the filename and format when prompted by a dialog box.

Printing the current crossplot/histogram

Select the **File/Print...** menu command. The **Print Preview** dialog box will appear, allowing you to set up the printing parameters and preview the image to be printed:



The left part of the window is the print preview area. The crossplot boundaries are shown in short dashes, the sheet boundaries – in long dashes.

The right part of the window contains additional printing parameters:

- **mm per unit on X axis** – horizontal scale in mm per unit on the X axis.
- **mm per unit on Y axis** – vertical scale in mm per unit on the Y axis.
- **Fit to page –** scale the image to fit to page.
- **xAxis Width (mm)** – total horizontal axis width with labels in mm.
- **yAxis Width (mm)** – total vertical axis width with labels in mm.
- **Print Setup –** allows selecting a printer and setting all necessary parameters in a standard Windows OS print setup dialog box.

- **Recalculate** – redraw the current image in the print preview area and recalculate the printing parameters taking the changes made in the dialog box into account.

The current printer settings are shown in the text area in the lower right part of the dialog box.

Working with active crossplots/histograms

An active crossplot (or histogram) is a user-selected crossplot/histogram on which polygons (intervals) determining the colors of points on other crossplots can be defined.

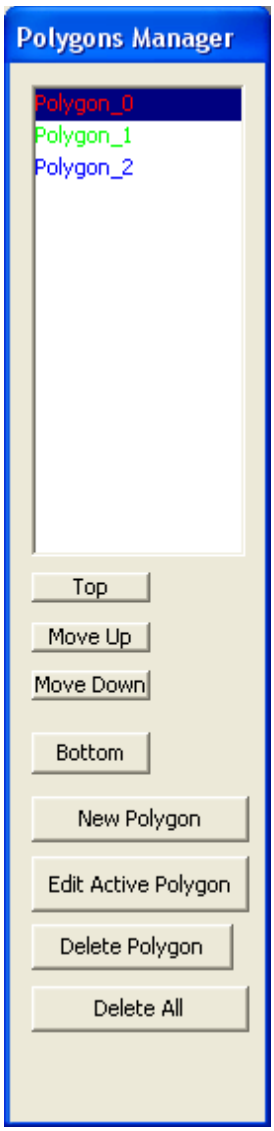
Only one crossplot/histogram can be active at any point of time. However, you can switch between several crossplots, making them active one at a time. In this case each crossplot will have its own set of polygons, as can be seen from the pictures below which show one and the same pair of crossplots twice – first with the right crossplot as the active one, and then with the left crossplot in that role:

You can make a crossplot/histogram active in one of two ways:

- By pressing the button on the toolbar or selecting the **File/Active plot** menu item in the crossplot/histogram window.

- By double-clicking the right mouse button on the crossplot/histogram name in the list in the Crossplot Manager window.

When a crossplot becomes active, the **Polygons Manager** window appears, allowing you to work with polygons/intervals.



The **Polygons Manager** window allows creating polygons/intervals on the active crossplot/histogram. It lists the names of created polygons or intervals. To select a polygon, click its name in the list. After that you can edit the selected polygon, change its properties and location relative to other polygons, or delete it.

When a polygon/interval is defined on the active crossplot/histogram, the points on all other crossplots corresponding to seismograms that fall within the specific polygon/interval assume the color of that polygon/interval. If polygons intersect, the first polygon on the list in the **Polygons manager** window will be shown on top of all other polygons on the active crossplot, and its color will be assigned to all points within the intersection area.

To move the selected polygon/interval up or down relative to other polygons/intervals, use the

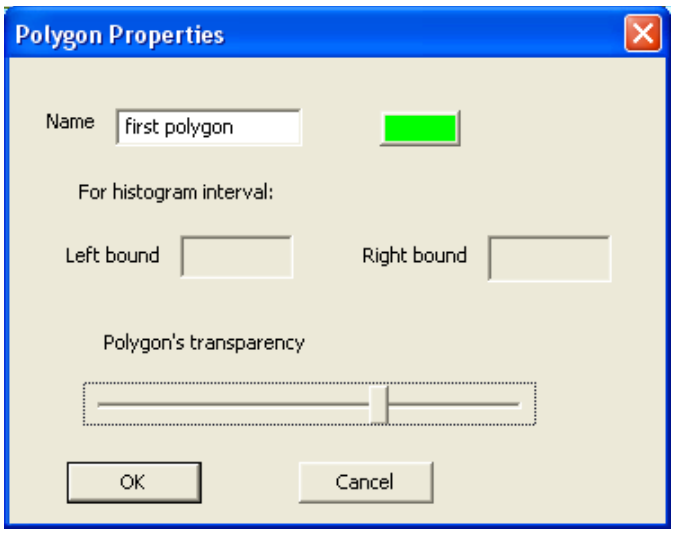
following buttons:

- **Top** – move the selected object to the top line of the list.
- **Move up** – move the selected object one line up.
- **Move Down** – move the selected object one line down.
- **Bottom** – move the selected object to the bottom line of the list.

The window's other buttons allow creating new polygons (**New Polygon**), editing the selected polygon boundaries (**Edit Active Polygon**), deleting the selected polygon (**Delete Active Polygon**) or deleting all polygons from the list (**Delete All**). A more detailed description of the main functions available when working with polygons/intervals is presented below..

Creating a new polygon/interval (the New Polygon/New Interval button in the Polygons Manager window)

To create a new polygon, press the **New Polygon** button in the **Polygons Manager** window. You can also do it by using the **Select/New Poligon(Interval's)** menu item in the active crossplot window or pressing the button on its toolbar. When you select this command, the **Polygon Properties** dialog box will appear:



Specify the polygon name in the **Name** field**.** (by default, the polygon_N name is assigned).

Select the polygon color using the color selection button .

In the histogram mode, the **histogram interval** parameters become available**:**

- **Left bound** – this field is used to set the interval's left boundary value in the current horizontal axis measurement units.

- **Right bound** – this field is used to set the interval's right boundary value in the current horizontal axis measurement units.

- **Poligon's transparency –** sets the polygon/interval transparency. The extreme right position of the slider corresponds to a fully opaque image, the extreme left – to a fully transparent one (in the latter case non-active polygons/intervals are not shown at all, and the active polygon/interval has only its outline shown).

After creating a new polygon, draw it in the active crossplot window by adding points with single clicks of the left mouse button. To close the path (complete the polygon), double-click the left mouse button.

To create a new interval in the active histogram window, select it by pressing the left mouse button on one of the interval's boundaries, dragging the cursor to the other boundary while holding the mouse button down, and releasing the mouse button.

Editing polygon/interval properties

To edit the name, color and transparency of the selected polygon/interval as well as interval boundaries, press the button on the toolbar or select the **Select/Active Polygon's Properties/Active Interval's Properties** menu item**.** This will open the already familiar **Polygon Properties** dialog box described above. The same can be achieved by double-clicking the left mouse button on the polygon/interval name in the list in the **Polygons Manager** window.

Editing polygon/interval boundaries

The geometry of polygons/intervals can be edited. To enter the active polygon/interval editing mode, press the **Edit Active Polygon/Interval** button in the **Polygons Manager** window. You can also do it by using the **Select/Edit ActivePoligon(Interval's)** menu item in the active crossplot window or pressing the button on its toolbar.

- In the editing mode you can ***move an active polygon node*** by holding and dragging it with the right mouse button.

- To ***add a new node,*** single-click the left mouse button.

- To ***delete an active polygon node*****,** double-click the right mouse button on that node.

- You can also ***move the entire polygon around*** using the right mouse button while holding down the Shift key.

Editing an active interval on a histogram is done in a similar way.

Deleting polygons/intervals

To delete a single polygon/interval, select it from the list and press the **Delete Polygon/Interval** button.

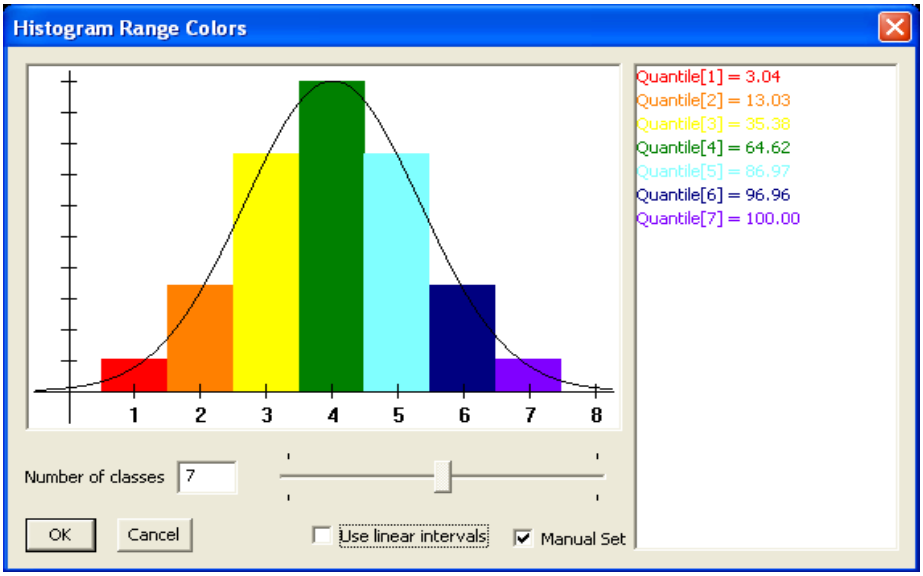
To delete all polygons/intervals on the active crossplot/histogram, press the **Delete All** button.

Automatic splitting of histograms into intervals (by quantiles or uniformly)

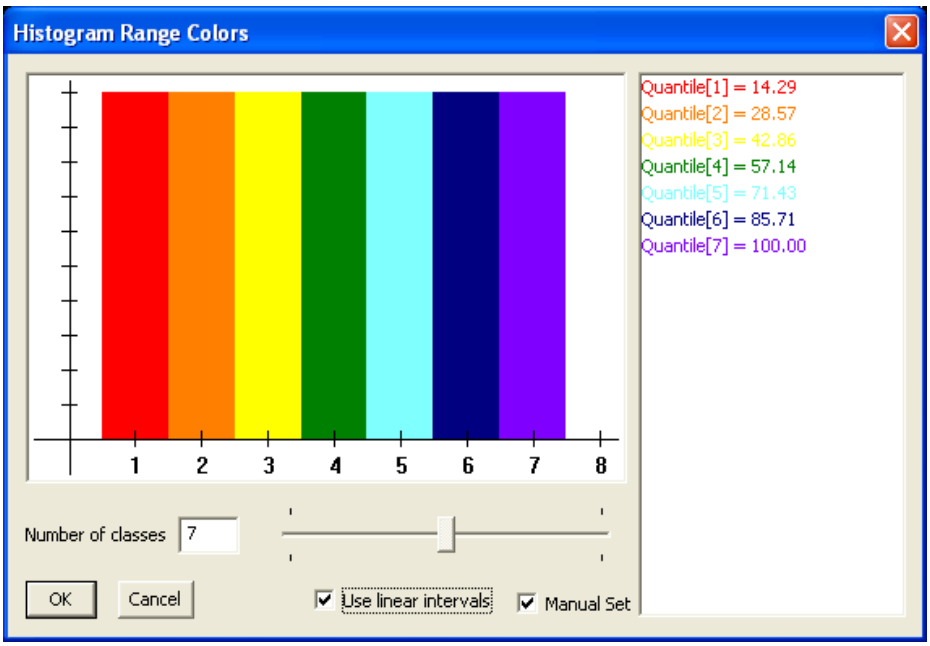
When working with histograms, you can choose to define intervals automatically. Intervals may be defined by quantiles (assuming normal distribution of the value in question) or linearly (assuming uniform distribution of the value in question). To define intervals automatically, use the **Select/Auto add intervals** menu item.

Selecting this command will open the **Histogram Range Colors** dialog box which will show the histogram broken down into 7 quantiles by default. The quantiles are calculated based on the assumption of Gaussian distribution of the header field values, although this may not actually be the case. The slider allows adjusting the shape of the Gaussian curve.

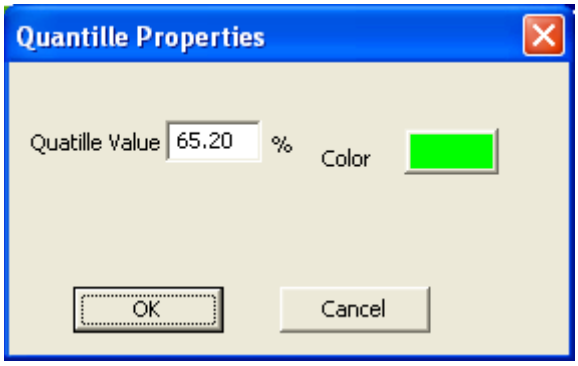
The dialog box is divided into two parts: the left one contains a graphic representation of automatic splitting into intervals, while the right one displays the current list of intervals to be created with the right boundary of each interval shown in percent.



To split the histogram into uniform linear intervals, select the **Use linear intervals** option**.**



You can specify an arbitrary number of quantiles/linear intervals in the **Number of classes** field (to update the interval diagram, click the mouse anywhere within the dialog box except for the field itself).



The **Manual Set** option enables/disables automatic definition of quantile values.

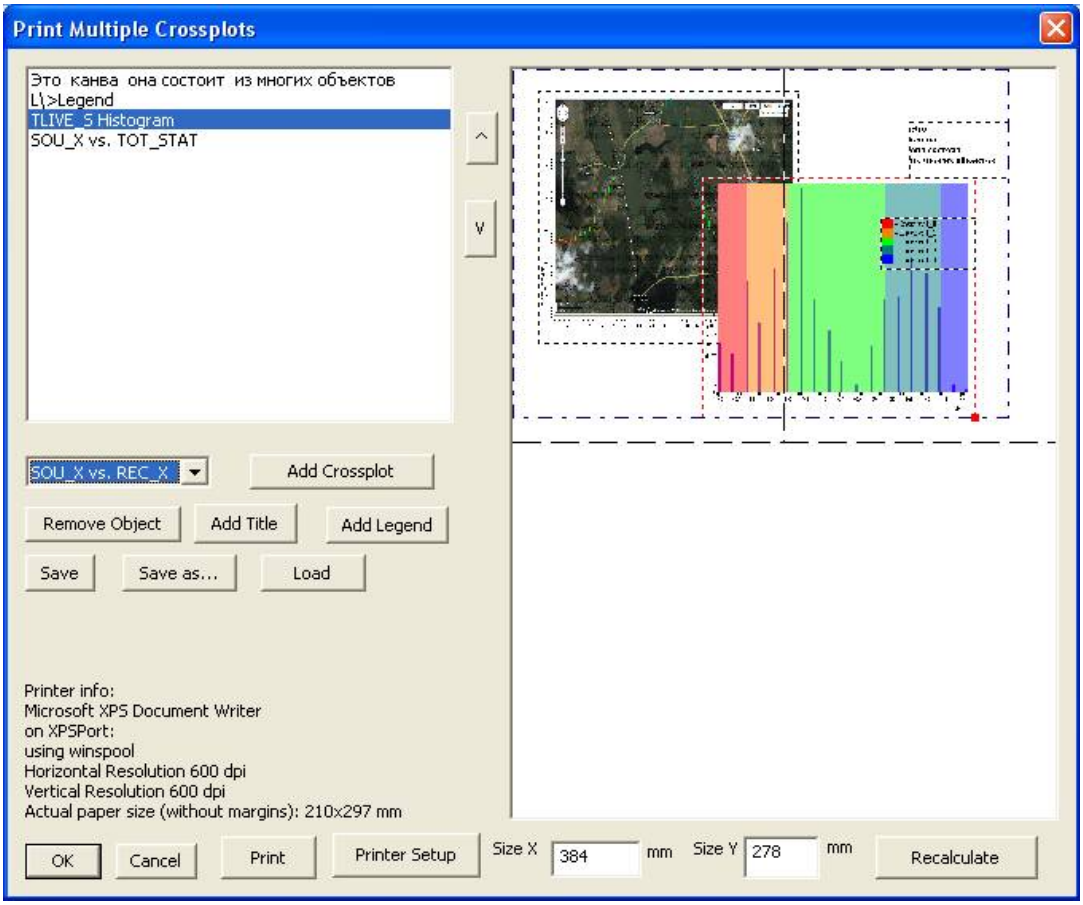
The right part of the window contains the list of quantiles and their values. The name of each quantile is shown in the same color as the corresponding interval on the histogram. Double-clicking a quantile name in the list opens the **Quantille Properties** dialog box, allowing the user to edit the quantile.

Quantille Value – quantile (right boundary of the interval) value in percent (this field is active only if the **Manual Set** box is checked in the **Histogram Range Colors** window).

The **Color** button is used to set the interval color.

Working with “canvases” – printing sets of crossplots/histograms with additional information

To start working with a canvas, press the **Canvas** button in the **CrossPlot Manager** window. The **Print Multiple Crossplots** dialog box will appear.

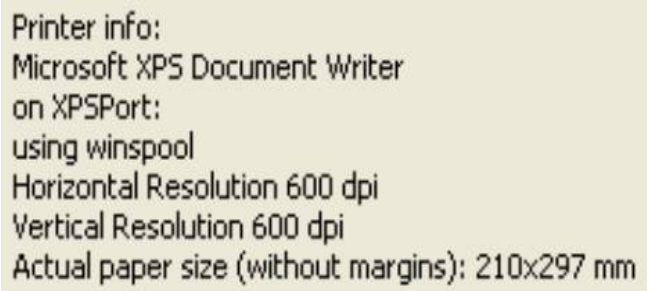


The **Print Multiple Crossplots** dialog box consists of the parameter setting area on the left, and the area where you can preview and interactively edit the canvas (which is a combination of objects – crossplots, histograms, legends and titles) on the right.

The top left corner of the dialog box contains the list of objects that have been added to the canvas and will be sent to the printer. Objects can overlap, so the order in which they are listed determines the order in which they will appear when printed (the first object in the list will be printed over all other objects etc.). You can change the order of objects using the up/down arrows to the right of the list.

After adding objects to the canvas and setting up their parameters, position and size (see below for a

detailed description of how to do this), you can print the canvas by pressing the **Print** button. The information on the currently selected printer and its settings will be displayed in the lower right part of the dialog box:



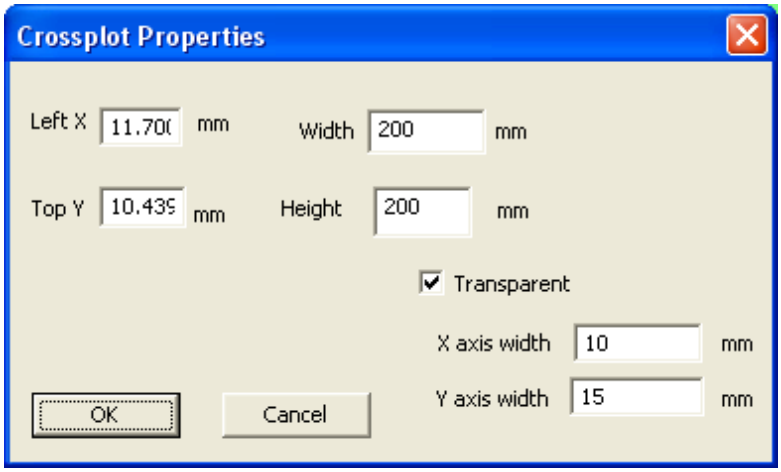
Press **Printer Setup** button to select a printer and set up its parameters.

You can save the current canvas state as the default collection canvas by pressing the **Save** button. This way, when you close the canvas and then press the **Canvas** button in the **CrossPlot Manager** window once again, the saved canvas will open.

If you need several different canvases, you can save the current canvas as a file on the disk (the **Save as** button) and later load it from that file (the **Load** button).

Adding a crossplot/histogram to the canvas

To add a crossplot or a histogram to the canvas and have it displayed in the preview area, select the object you need from the drop-down list in the left part of the **Print Multiple Crossplots** dialog box and press the **Add Crossplot** button. The **Crossplot Properties** dialog box will appear.

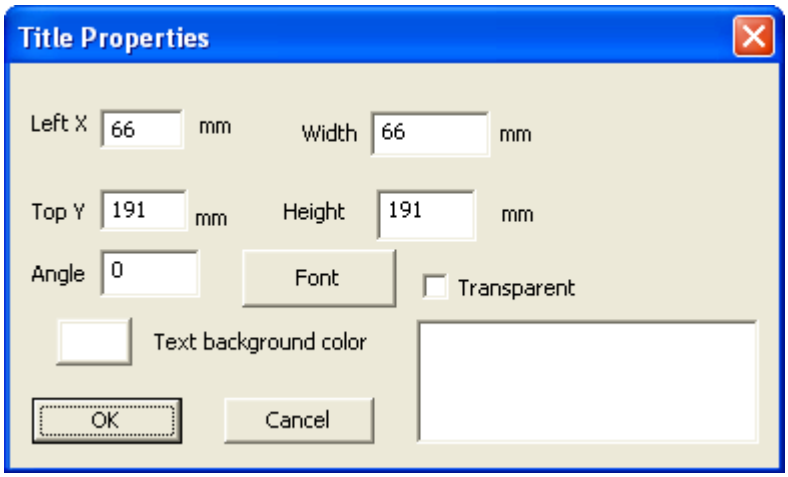


- **Left X –** X coordinate of the image top left corner in mm.
- **Top Y –** Y coordinate of the image top left corner in mm.
- **Width –** image width in mm.
- **Height –** image height in mm.
- **Transparent** – enable/disable image background transparency.
- **X axis width** – X axis width in mm.
- **Y axis width** – Y axis width in mm.

The object size and position on the canvas specified in this dialog box can later be changed interactively in the preview area. When you are done setting the parameters, press **OK**. The selected crossplot or histogram will be added to the canvas.

Adding supplementary information to the canvas

Besides crossplots and histograms, you can add supplementary information – titles and a legend – to the canvas.

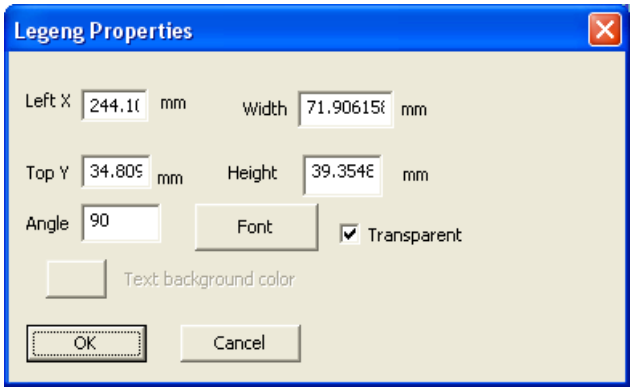


Add Title – add a title to the canvas. Pressing this button opens the **Title Properties** dialog box.

- **Left X –** X coordinate of the title top left corner in mm.
- **Top Y –** Y coordinate of the title top left corner in mm.
- **Width** – title width in mm.
- **Height –** title height in mm.
- **Transparent** – enable/disable title background transparency.
- **Angle** – title rotation angle in degrees in the 0-90 range.
- **Font** – title font setup.
- **Text background color** – text background color.

The object size and position on the canvas specified in this dialog box can later be changed interactively in the preview area.

A legend is a list of polygons within the active crossplot or intervals within the active histogram which define the colors of points on all other crossplots. Press the **Add Legend** button to add a legend to the list of objects displayed in the preview area. The following dialog box will appear:



- **Left X –** X coordinate of the title top left corner in mm.
- **Top Y –** Y coordinate of the title top left corner in mm.

- **Width –** title width in mm.
- **Height –** title height in mm.
- **Transparent** – enable/disable title background transparency.
- **Angle** – title rotation angle in degrees in the 0-90 range.
- **Font** – title font setup.
- **Save as –** save the canvas parameters to a text file with the specified name.
- **Save** – save the changes in the canvas parameters to a text file with the default name.
- **Load –** load canvas parameters from a text file.

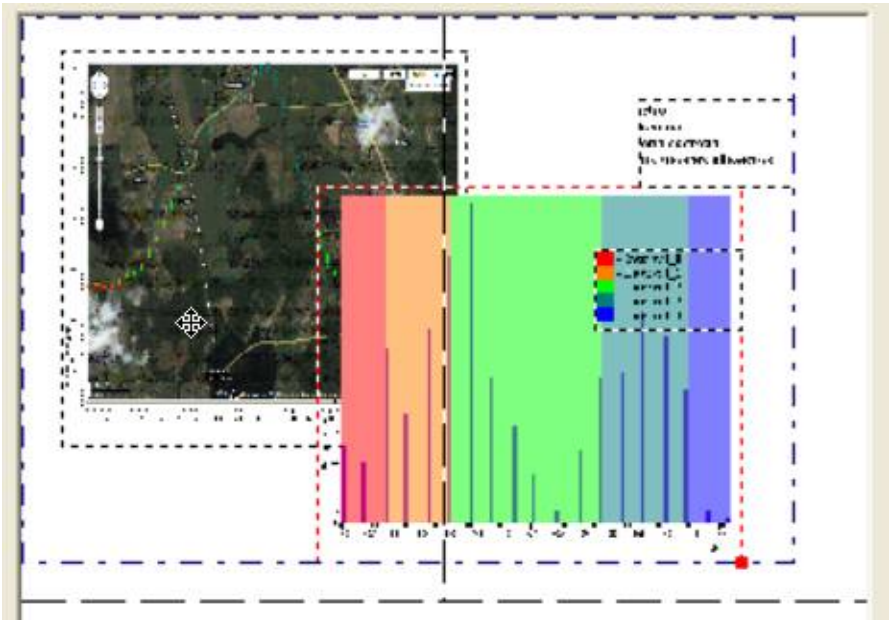
The object size and position on the canvas specified in this dialog box can later be changed interactively in the preview area.

Deleting a crossplot/histogram from the canvas

To delete a crossplot/histogram from the preview area, select its name in the list and press the **Remove Object** button.

Working with the canvas preview and editing area

The canvas preview and editing area allows you to interactively change the position and size of objects on the canvas.



- **Selecting active objects.** Select an object by left-clicking on it while holding down the Ctrl key. In places where several objects overlap each other the topmost object will be selected (in accordance with the order in the list of displayed objects). You can also select an object by left-clicking on its name in the list (in the left part of the **Print Multiple Crossplots** dialog box).

- **Moving objects around the canvas.** Place the mouse cursor over the selected object you want to move. The cursor will change its appearance to . Then grab the object with the left mouse button, move it to the desired location, and release the mouse button.

- **Resizing objects.** Place the mouse cursor over the lower right corner of the selected object (the selected object will have that corner marked with a red square), press the left mouse button, move the cursor to the new location while holding the mouse button down, and release the mouse button. The object corner will move with the cursor, and the object size will change accordingly.

- **Changing object parameters.** To open the object parameter setup dialog box, double-click the object with the left mouse button.

When objects are moved or resized on the canvas, the lines showing sheet layout and image printing borders on the canvas are automatically redrawn.

3C Processing (multi-component processing)

Asymptotic CCP Binning

Asymptotic Common Conversion Point Binning module is designed for performing asymptotic binning of converted PS waves' data. Header values, relating to the common conversion point (CCP) are filled in the data when the module is run within the flow: CCP_X, CCP_Y and CCP.

Moreover, the module can be used for conventional CDP binning in 2D case.

When using the module for CDP binning, you should indicate the value **Gamma** = 1. Then, just after this module in the flow place a **Trace Header Math** and write down the following expressions there:

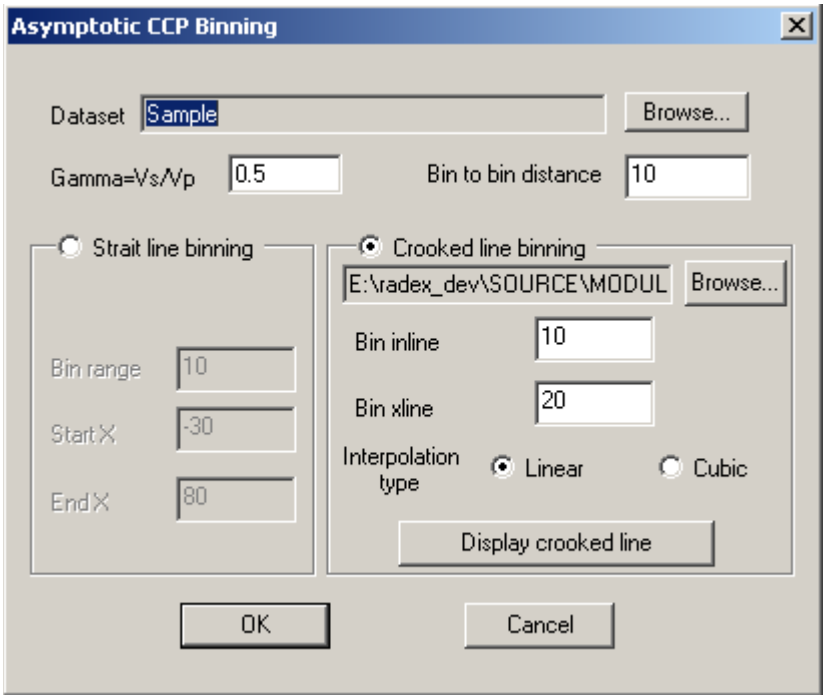
CDP_X = [CCP_X]

CDP_Y = [CCP_Y]

CDP = [CCP]

Module parameters dialog

The following dialog box appears while accessing the module:



- **Dataset** field is intended for selection of traces' set by the user for binning procedure. A window for selection of dataset from the project database appears when you click the button **Browse…**.

- **Gamma=Vs/Vp** field is intended for input of shear waves' velocity to compressional waves' velocity ratio, which is considered to be a binning parameter (in case you use the module for CDP binning, you should specify 1).

- **Bin to bin distance** is the profiling parameter and corresponds to the distance between bins in meters, calculated on binning profile.

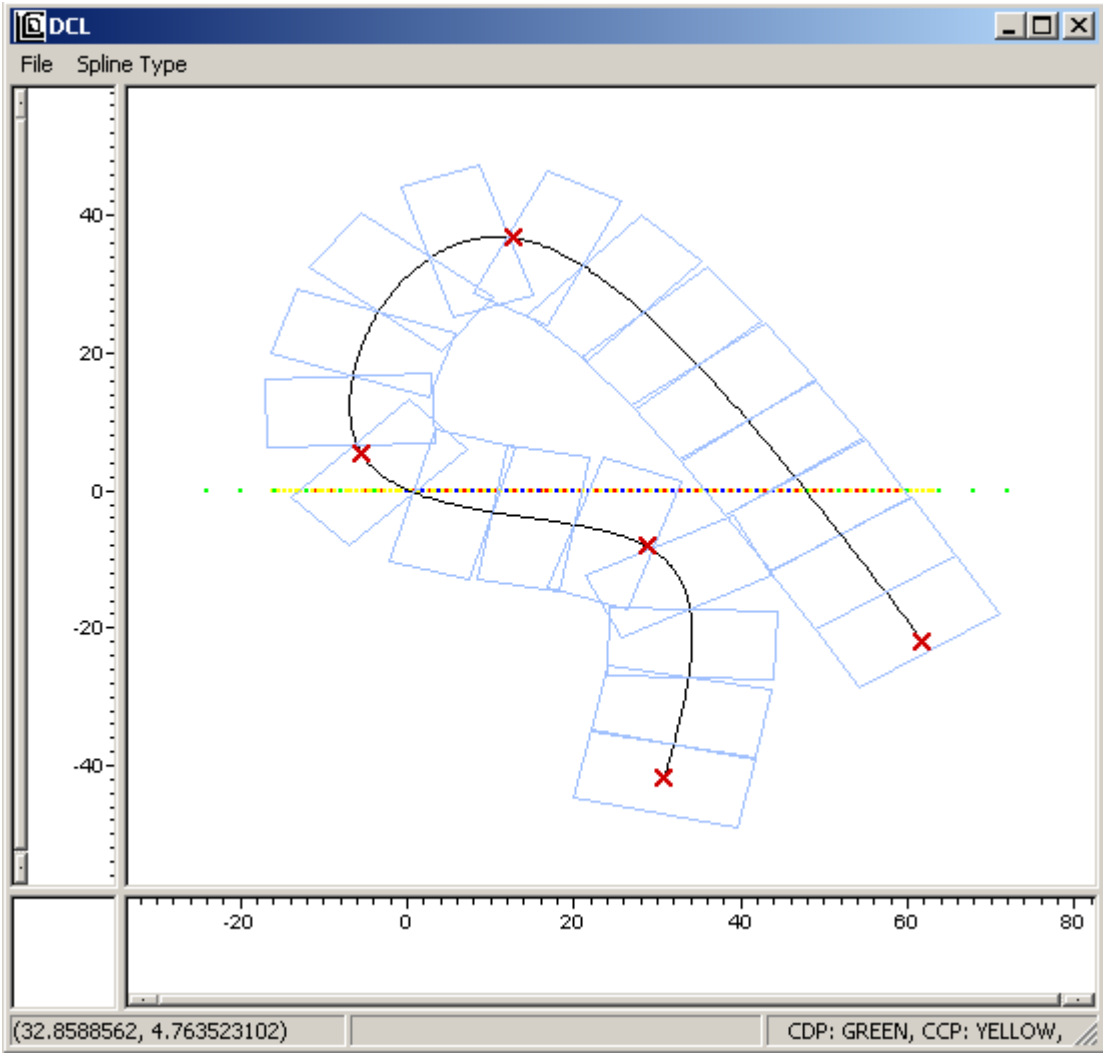
- Option **Strait line binning/Crooked line binning** allows selecting the binning type: one-dimensional or two-dimensional. In the first case (when you have chosen **Strait line binning**) it is supposed that all traces for binning have the same REC_Y and SOU_Y and the binning profile is specified by the segment [**Start X, End X**]. **Bin range** parameter specifies here the bin width along the profile.

- In the second case (**Crooked line binning** is on) the profile is specified by a crooked line, read from the file. **Browse…** button of the section allows selecting the filename of the profile, while the button **Display crooked line** – visualizing and editing a profile along with a crooked line of bin profiling, receivers, sources, CDP and CCP. If the file has not been specified, a straight profile will be generated, connecting the points of source and receiver of the first trace. **Bin inline** and **Bin**

xline parameters allow specifying linear dimensions of bins: bin width along the binning curve and height across the curve, correspondingly.

Visualization /curve editing window

A visualization/profile curve editing window appears when you click the button **Display crooked line**:



Scale bars of the window are situated at the left and at the right sides of the screen. The dimensions of windows for these bars can be changed using mouse buttons. Except scale indication, a selected fragment of an image can be changed using these scales.

To magnify the image you have to indicate on a corresponding bar the start coordinate of the fragment to be magnified via the left mouse button (**MB1**), and holding left mouse button, drag the

mouse pointer to the end coordinate. The selected fragment will be magnified and scaled to fit the viewport.

The viewport depicts:

- in black – a profiling curve,
- in green and yellow – CDP and CCP,
- in blue and red – receivers and sources.

Red crosses denote the nodes of profiling curve.

To change the position of the existing node, grab it with the left mouse button (MB1) (the cross will change its color), drag it to a new position and release a button. The node will shift to the position under the cursor. At the same time the bins will be redistributed.

To add a node to a curve you have to click a mouse button twice. A node will be added to the nearest point of the curve.

To delete the node, click on it with the left mouse button (MB1), holding simultaneously **Ctrl** button.

The status bar reflects: at the left side – metric coordinates, corresponding to the cursor position, at the right side – prompting message on the colors of visualizing objects.

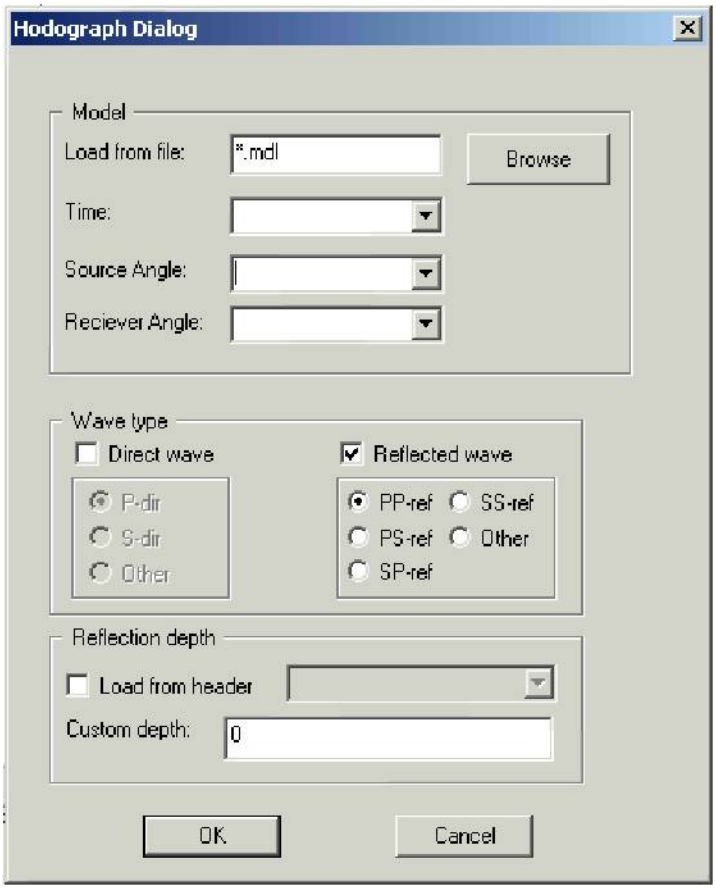
Modeling

Hodograph

This module allows calculation of arrival time of direct wave or wave reflected from specified reflecting boundary on the assumption of horizontal layered medium. The medium model should be specified as a text file. The depth of reflecting boundary should be specified separately.

The seismograms running in the flow with filled in fields of source and receiver coordinates are input into the module. In the module the wave arrival time, the angle of incident wave at which it departs from the source and the angle at which the reflected wave comes to the receiver are calculated for every trace (for source-receiver position indicated in its headers). These values are recorded into indicated trace header fields.

When this module is activated the following window appears:



Horizontal layered model must be specified in the text file. To select the file in the **Model** group of parameters, click the **Browse...** button and choose the file in the standard dialog box. Besides, the file name can be entered manually in the **Load from file** field.

The file must be a text one and contain the values presented as a table with blanks or tabs as separators. The first line should be the line that starts with *~A* symbols and then contain the names of the columns available in the file:

- Layer bottom depth *(Z)*
- P-wave velocity in the layer *(Vlay)*
- S-wave velocity in the layer *(Vslay)*
- The type of wave in the layer *(WaveType)*. To identify the PP-wave in the layer the 0 constant is used, the PS-wave - 1, SP-wave - 2, SS-wave - 3. When specifying the type of direct wave only 0 (P-wave) and 3 (S-wave) constants are used.

After that, for every layer the respective values should be specified in the order similar to columns headers position. The depths are specified in meters, velocities - in km/s.

An example of table describing the model:

~A Z Vlay Vslay WaveType 10

1.6 0.8 0 2000 3.2

1.791 1 2540 2.6 1.4 2

3000 3.51 1.54 2

The *WaveType* column allows the user to assign the type of wave individually for every layer of the model. The module uses the values from this column only if in the module parameters dialog box the type of the wave is specified as **Other.** Otherwise, if the type of the wave explicitly indicated in the module parameters it is considered as same for all layers and the values in the *WaveType* columns are disregarded.

When the file with the model is specified, select the type of the wave for which the hodograph will be calculated. To do this, use the **Wave type** group of parameters available in the module dialog box. Here, activate either **Direct wave** option for direct wave or **Reflected Wave** option

for reflected wave.

The direct wave can be of the following types:

- **P-dir** - direct P-wave;
- **S-dir** - direct S-wave;
- **Other.** When this option is activated the type of the wave is taken from the text file describing the lay model.

The reflected wave can of the following types:

- **PP-ref** - PP-type wave (both the incident wave and reflected waves pass through all layers as the P-wave);
- **PS-ref** - PS-type wave (the incident wave passes through all layers just as the P-wave but the reflected wave passes through all layers as the S-wave);
- **SP-ref** - SP-type wave (the incident wave passes through all layers just as the S-wave but the reflected wave passes through all layers as the P-wave);
- **SS-ref** - SS-type wave (both the incident wave and reflected waves pass through all layers as the S-wave);
- **Other.** When this option is selected the types distribution in layers of the model is taken from the text file describing the lay model.

For reflected wave, specify the depth of the boundary (in meters). The time arrival of reflected from this boundary waves will be calculated. The depth of reflecting boundary should be specified in the **Reflection depth** field. It can be specified by one of the two methods:

- In the **Custom depth** field you can set the reflection depth manually for all source- receiver pairs at the same time.
- Activate the **Load from header** option and set the reflection depth previously saved in the headers field. To do this, select the desired header from the list.

The module calculates the rime of wave arrival from specified reflecting boundary, the angle of incident wave at which it departs from the source and the angle at which the reflected wave comes to the receiver. These values are recorded into indicated trace header fields. The header can be selected in the **Model** group of parameters:

- **Time** - from the list, select the header field where the time of wave arrival will be recorded;

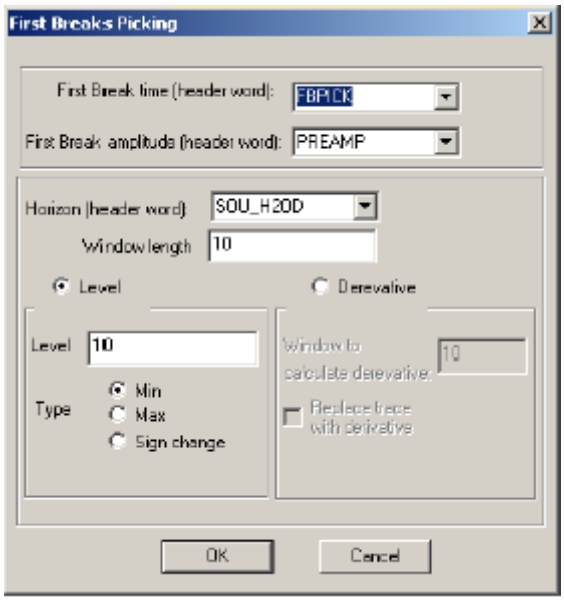
- **Source Angle** - from the list, select the header field where the angle of incident wave at which it departs from the source will be recorded;

- **Receiver Angle** - from the list, select the header field where the angle at which the reflected wave comes to the receiver will be recorded.

Auto Picking

First Breaks Picking

The module is designed for automatic detection of first breaks. The figure represents a dialog of module options selection. The traces with time marks in headers, starting from which the first breaks detection will be performed (**Horizon (header word)**) are input; as a result two header fields (**First break time**) with first break times and amplitudes on traces corresponding to those time marks (**First break amplitude**) are filled in. The tracing of first break time is performed in window with the length **Window length** ms.



Two algorithms are implemented:

- Evaluation of first breaks by trashold level (**Level**)

The search of first sample, which amplitude exceeds (if the parameter **Level** is positive) or less than (if the parameter **Level** is negative) **Level**, is implemented in this mode. The first characteristic point of the indicated **Type** (minimum, maximum or zero crossing) is searched further on starting from this time mark. The time of the characteristic point found is considered to be the first break time.

- Evaluation of first breaks from the maximum of energy growth (**Derivative**). Mean energy is calculated in two windows, with the length **Window to calculate derivative**, situated in series, the relation of energy in the second window to the energy in the first window is attributed to the central part of a window.

The first break time is considered to be the maximum of the relationship. We know from the experience that good window length is equal to impulse duration (you can decrease the window length).

The **Replace trace with derivative** flag helps to substitute the trace for the described function that can be useful for parameters testing.

Marine

Zero-Offset DeMultiple

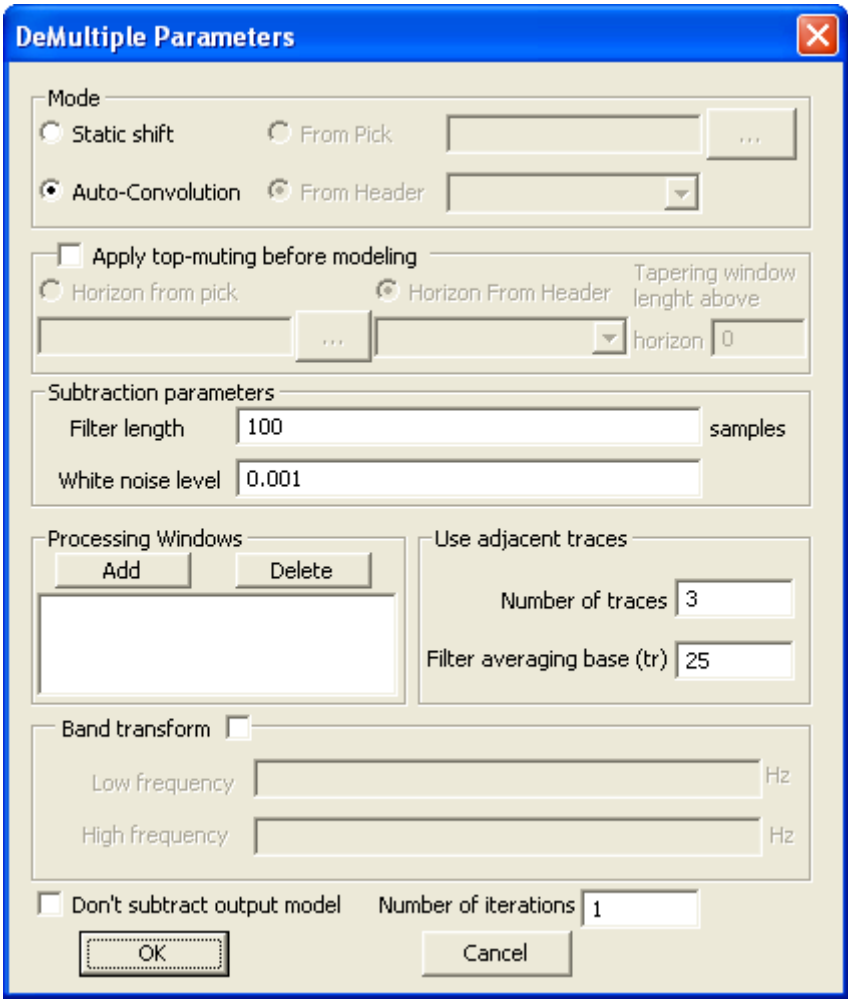
The module is deigned for demultiple of near-offset single-channel or stacked seismic data. The algorithm is based on adaptive subtraction of a model of multiples from the original wave field. The model is obtained from the data itself, either by static shift of the original traces of by auto-convolution. The adaptive subtraction algorithm used here is the same as implemented in the *Wave Field Subtraction* module and is discussed in more detail in the part of the Manual dedicated to that module.

In general terms, a special filter is calculated for each trace basing on both the original data traces and the model traces. This filter, when applied to the trace is trying to minimize the RMS amplitudes of whatever is found similar between the trace and the model. When the filter is

calculated, it is accounted for non-stationarity to make it adaptive to the events that are quite similar but *not exactly* the same. This makes the subtraction to a certain extend efficient even for rather approximate models, when the arrival time or/and the amplitude of the modeled multiple differ from the actual observation. However, the more similar is the model of multiples to the real multiples observed, the more efficient is the subtraction. For this reason, the shorter is the source-receiver offset, the better is the result that can be obtained, because the most accurate multiple modeling can be easily made on zero-offset data. For the same reason, the module is less efficient if the data is significantly disturbed by sea swelling because the primaries and the multiples are disturbed differently and the resulting model becomes less similar to the observation.

When the filter is calculated for a particular trace of the original data, beside the corresponding trace of the model some adjacent traces of the model can also be used. It might happen that some extra similarity to what is observed on the original data trace can be found there. Thus, using adjacent traces could make the subtraction effect stronger. On the other hand, the filter shall not differ too much from trace to trace. For this reason, filters calculated for each of the original traces can afterwards be averaged over the base of several traces. This would make the subtraction milder and help to avoid the effect of erasing a gap around the removed multiple.

When the module is activated the following dialog appears:



- Mode group of parameters defines a way how a model of multiples will be generated:

Static shift – by shifting each trace to a specified time (typically, to the arrival time of the seafloor reflection). The static shift for each trace can be taken either **From Pick** in the database or from a specified trace header field (**From Header**) of each trace.

Auto-Convolution – when this option is selected, a result of auto-convolution of the trace is taken as the model of multiples. This mode require no extra parameters.

- **Apply top-muting before modeling** – when the option is checked the data will be top-muted before the model is generated. You can select either Horizon from pick or Horizon from header to define the muting. Here the Tapering window length above the horizon defines a window in ms above the horizon where the amplitudes will gradually fade out down to 0.

- **Subtraction parameters** define how the model will be subtracted from the data.

Filter length – the length of the adaptive filter that is used to form the result of the subtraction, specified in samples. The filter is calculated from both the original trace and the trace(s) of the

model and is aimed to find similarities and minimize their RMS amplitudes at the resulting trace. Typically, the filter shall be longer than the wavelet length. Normally until a certain extend the longer is the filter the stronger is the subtraction effect and the more time is required for the processing.

- **White noise level** – regularization parameter, the larger is the value the more subtle is the subtraction effect. More details are provided in the W*ave Field Subtraction* module description.

- **Processing windows** — the processing can be made independently is separate windows. Then, an individual subtracting filter is calculated for each window. This may lead to a more accurate result. You might wish to make a separate window for each strong seafloor multiple observed.

The windows are defined by picks – each pick is considered to be a boundary between a window above it and another window below it. Thus, when no picks are specified in the list here the whole data range is considered as one window. One pick defines 2 windows (above and below), 2 picks define 3 windows (above both, in between of them, and below both), etc.

Using the buttons above the list of currently selected picks you can Add picks to the list and Remove them. You shall make sure that the picks added to the list do not criss-cross, otherwise the module behaves predictively. It is not recommended to make the windows too narrow, otherwise it may lead to erasing everything there.

When several processing windows are used, the Filter length and White noise level (as well as the Band transform parameters discussed below) are to be indicated for each window individually. Values corresponding to different windows are separated by colon (:). When a new pick is added to the list, the parameter strings are adjusted automatically by duplicating the last parameter. If you wish to exclude some window from the processing, set its filter length to 0.

- **Use adjacent traces** – this group of parameters regulate use of adjacent traces of the model for filter calculation as well as averaging of the resulting filters over the base of several traces.

- **Number of traces** – specify the number of adjacent traces of the model taken on each side of the current trace to be used for filter calculation. When this parameter is 0, the filter for the current trace is calculated using the corresponding trace of the model only. When it is set to 1, the filter is calculated basing on 3 traces of the model: the current traces and 1 extra adjacent

trace each side, etc. (For more details of the implementation refer to the *Wave Field Subtraction* module description). Using more traces here would lead to stronger subtraction and longer processing time. A good starting guess for this parameter is 3. Then you may try to increase the number try and subtract more.

Filter averaging base – specify here how many adjacent traces the resulting filters will be averaged over. Normally, relatively big values are good here. You can start with 25 and then see if you would like to make subtraction stronger (smaller number) or milder (larger number).

Band transform — this option limits the bandwidth of the operation, saving processing time and providing additional frequency filtering at the same time. Specify **Low frequency** and **High frequency** to be processed for each of the processing windows.

Number of iterations – sometimes it is worth making several iterations of adaptive subtraction to achieve better result. At each iteration, the initial model of multiples is subtracted once again from the result of the previous iteration. When the first iteration removes the most high-energy part of the multiples only, additional iterations may help to get rid of the remaining lower-amplitude energy. However, each extra iteration requires additional processing time.

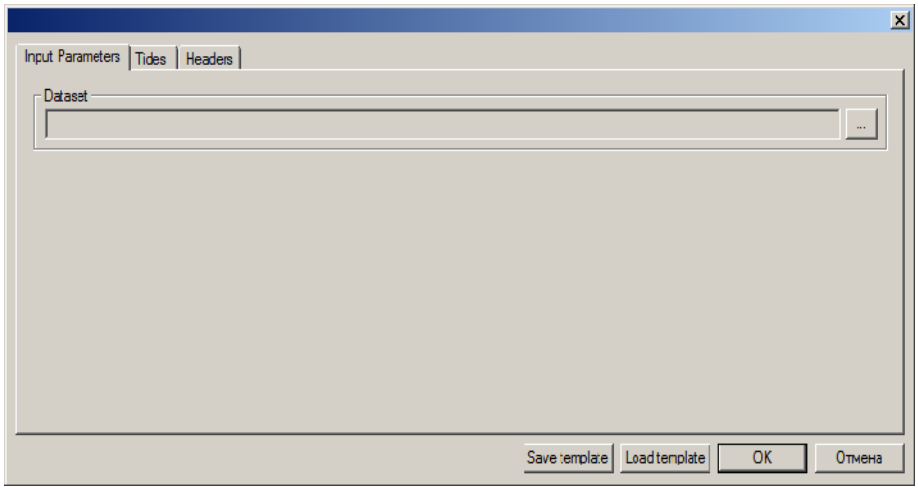
Don't substract output model – when this flag is checked, no subtraction of the model is performed. Instead, the module merely outputs the model of multiples prepared for subtraction. Often it is crucially important to see the model in order to understand the subtraction result (especially, when it happens to be not as good as was expected).

Tides Import*

The module imports static corrections for tides into trace headers of a dataset. The tides data are usually represented as shape of "Time-Sea level altitude" tables. In order to import this data correctly, it is necessery that all traces have date and time information in their headers. Then the values of the sea level altitude can be related to specific traces. The time spacing in table is usually regular but infrequent (typical intervals between measurements are 1-10 minutes). Since the shooting interval is normally much shorter, when the static corrections are loaded linear interpolation is applied.

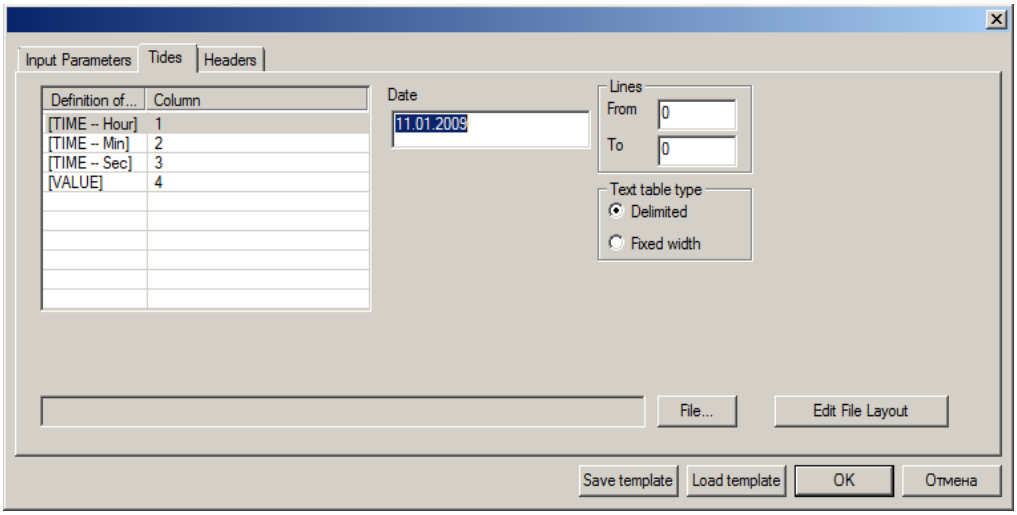
The **Tides Import** is a stand-alone module. Its parameter dialog window consists of several tabs:

Input parameters tab



Dataset – click the **...** button to the right of the string to select a dataset where the tidal corrections are to be imported to.

Tides Tab



On this tab select a text file with the sea level altitudes here and define its format.

Definition of Field

Positions/columns from which the corresponding values will be loaded are indicated.

Time positions (hour, minute, second) and the column with tide values to be imported (VALUE) are to be indicated.

In the field **Date** specify the date of the first record in file (since only hours, minutes and seconds are read from the file)

Lines sets of the range of required lines from the file.

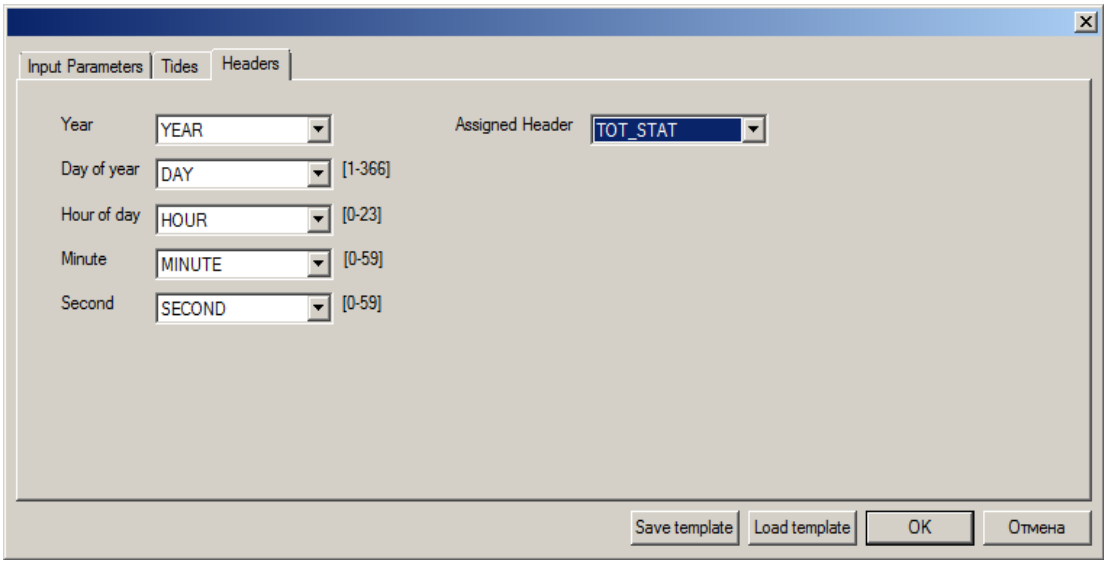
Text table type sets the file record type:

- **Delimited** -space-separated columns.
- **Fixed width** – field positions are specified by the offset from the start of the line.

The **File** button allows selecting the file with the information to be loaded.

Use **Edit File Layout** button to specify the structure of the tides file.

Headers Tab.



This tab sets necessary header fiels:

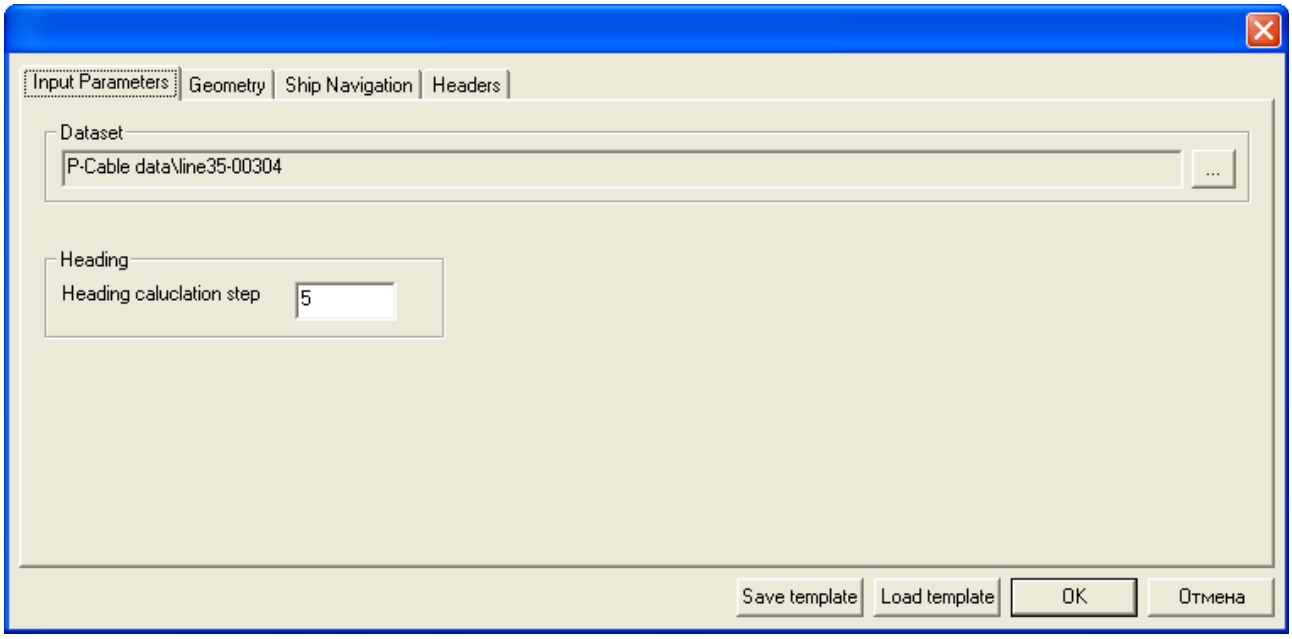
- fields with date and time.

- **Assigned Header –** sets a header word for static corrections.

Marine Geometry Input

The module is designed for calculating geometry (tie) of marine data, acquired via one multichannel cable, virtually with variable distance between receiving groups. The tie is calculated in terms of spread geometry and navigation text file, acquired by GPS receiver, positioned on the vessel. The parameters' dialogue contains several tabs.

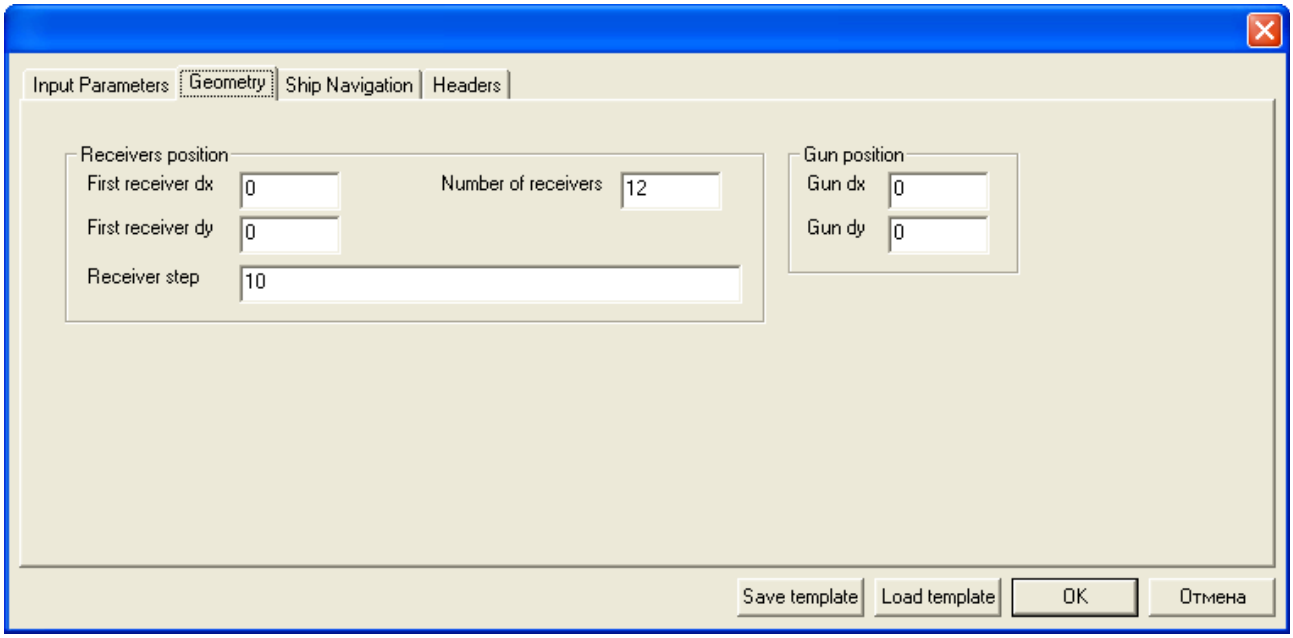
Input Parameters



Dataset – dataset in the project database, in which array geometry has to be allowed for.

Heading calculation step –step for points from navigation file for heading calculation at each instant of time. It is supposed that the cable is aligned along the heading line when calculating coordinates. When the step is equal to 1, the heading calculation will take into consideration every point, that due to navigation errors can lead to significant oscillations. Step increase is equivalent to navigation data smoothing and makes the heading calculation more viable.

Geometry



The spread parameters are specified.

First receiver dx

First receiver dy

Offset of the first receiver group with regard to GPS antenna position, the offset is specified in coordinate axes, associated with the vessel, while the origin coincides with GPS position, X-axis points to starboard, У-axis to stern.

Number of receivers - number of receiver groups in the spread.

Receiver step –step between the neighboring receiver groups along the cable. If the steps are different, the values are comma separated. If there is one number, the distance between groups is considered to be equal.

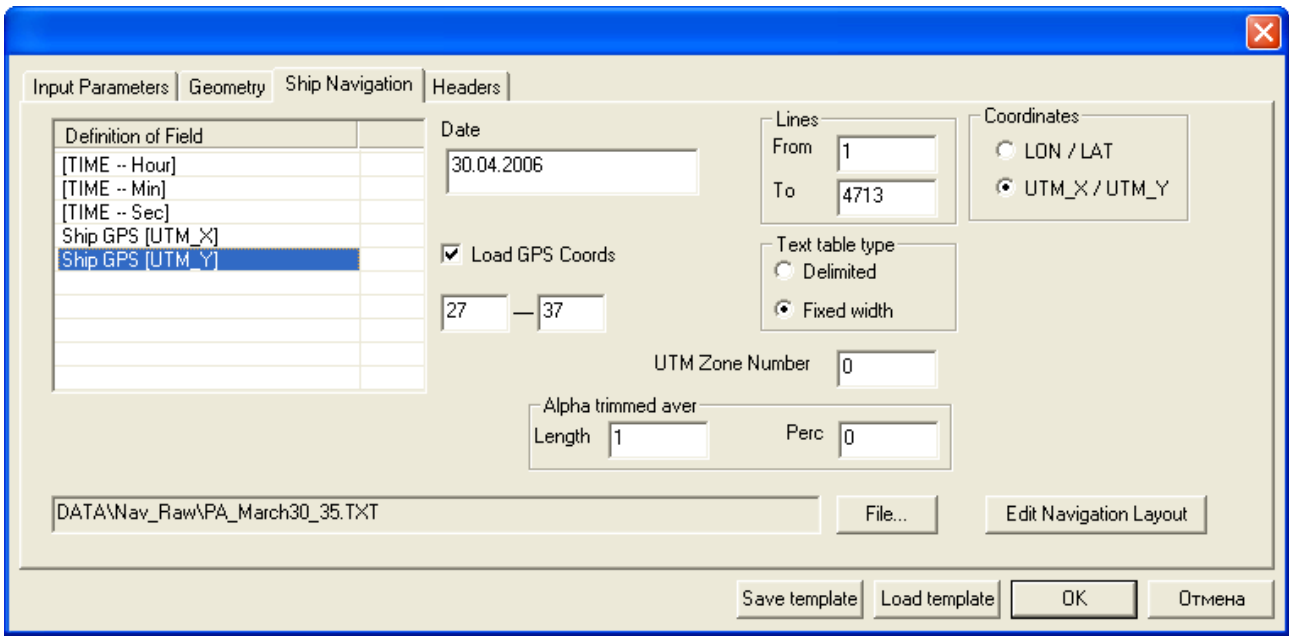
It is supposed that the cable is aligned along the heading line when calculating receiver coordinates.

Gun dx

Gun dy

Source offset in regard to GPS position. Offsets are specified similarly to the first receiver offset.

Ship navigation



Navigation file and information on the file structure is reported on this tab.

Definition of Field.

Positions/columns from which the corresponding values will be loaded are indicated.

Time positions (hour, minute, second) and coordinates (UTM_X/UTM_Y or LON/LAT depending on the switch ***Coordinates***) should be indicated correctly.

Field ***Date*** includes the date of the first read entry in the file (as only hours, minutes and seconds are read from the file).

Lines specifies the rows, read from the file.

Text table type specifies the file record format:

Delimited – space-separated columns.

Fixed width – field positions are specified by offset from the start of line.

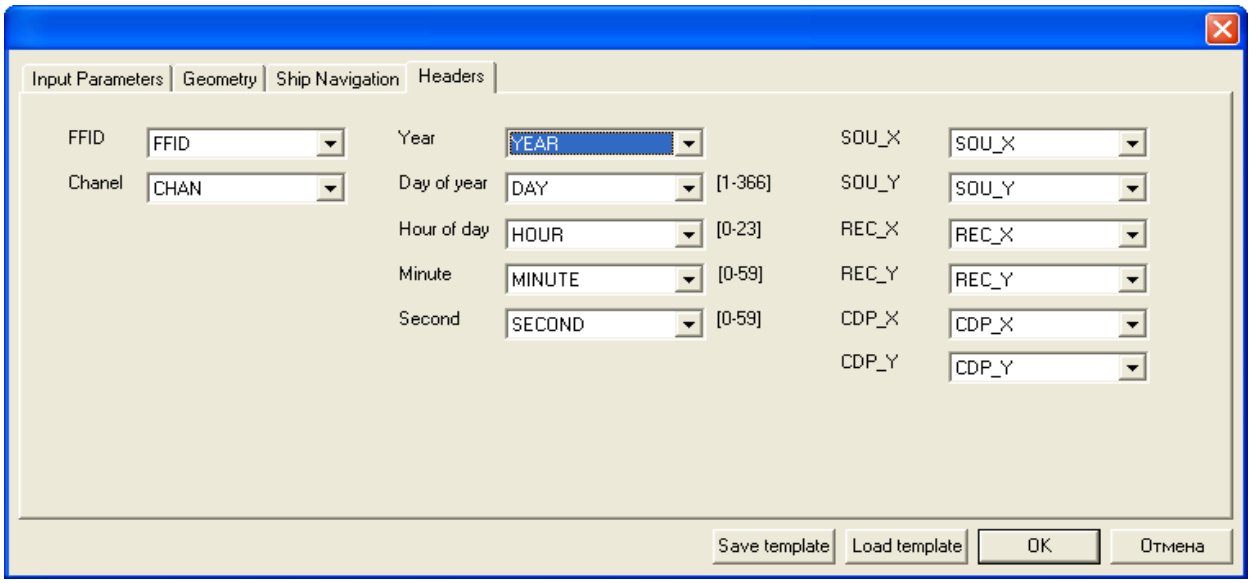
UTM Zone Number – indicates into which UTM area the coordinates will be recalculated when performing transformation LON/LAT → UTM_X/UTM_Y.

Alpha trimmed aver specifies the parameters of smoothing coordinates, loaded from the navigation file. Smoothing is carried out before assignment of coordinates to traces.

File button allows selecting navigation file.

You can call a tool of navigation file structure assignment using the button ***Edit Navigation Layout***.

Headers



All necessary header fields are indicated on this tab:

- Identification fields – FFID and CHAN.
- Fields where the time marks are stored
- Fields to which coordinates will be output

Deghosting* (Ghost wave suppression on near offsets)

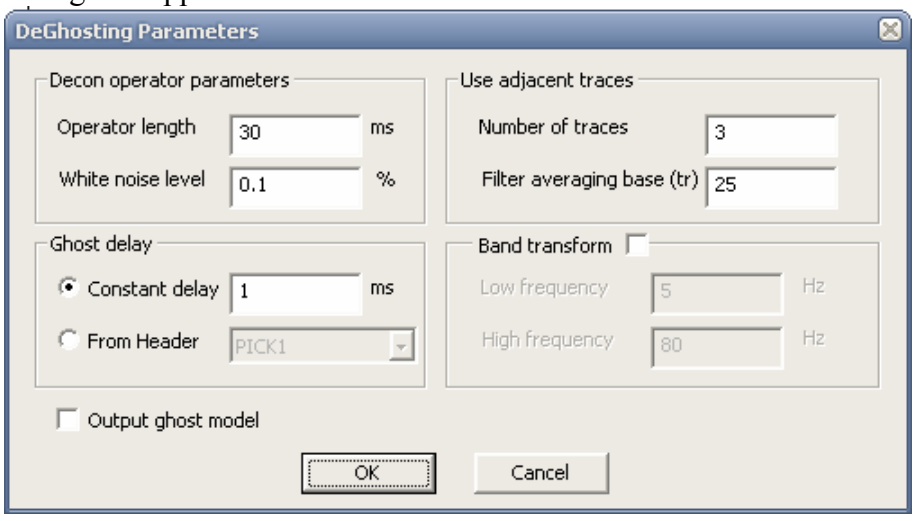
This module is used to suppress ghost waves in single-channel or stacked data obtained with small source-receiver offsets. The algorithm implemented in the module is based on adaptive subtraction of the ghost wave model from the source wave field. The ghost wave model is created from the data itself by means of static shifting of the original traces by the specified time. The ghost wave adaptive subtraction algorithm is similar to the one used in the Wave Field Subtraction module and is described in detail in the relevant section of the User Manual. The basics of the module operation are briefly described below.

A shaping filter that minimizes the results of the ghost wave model subtraction from the original field is constructed for each trace based on the original and model data. The filter searches for similar reflections present both in the original data and in the ghost wave model and minimized their root mean square amplitudes in the resulting field. Additional nonstationarity that allows the filter to adapt to events that are similar but not *identical* is introduced in the process of the filter calculation.

This allows the subtraction algorithm to work efficiently to a certain degree even with approximate ghost wave models wherein the arrival times and reflection amplitude differ somewhat from the observed ones. However, the closer the ghost wave model is to the actually observed ghost waves, the better it will be subtracted. Therefore, the best results can be achieved with minimum source-receiver distances, since zero offset allows obtaining the most accurate model. For the same reason the module works less efficiently when data are substantially affected by sea disturbance.

In the process of filter calculation for each individual original data trace several adjacent traces may be used in addition to the model trace corresponding to the original data trace, since it may be found that the specific features of the current trace are more similar to fragments of the adjacent traces in the model. Consequently, use of adjacent traces may result in stronger subtraction. On the other hand, filters generated for adjacent traces should not differ from each other substantially. Therefore, it may be useful to average the filters calculated for each trace with several adjacent traces. This operation results in a softer subtraction and allows avoiding the effect of a “hole” around the suppressed multiple reflection.

The following dialog box appears when the module is launched:



- **Decon operator parameters** – parameters of the subtraction operator
- **Operator length** – sets the shaping filter length in milliseconds. As a rule, the filter length should be equal to or greater than the wavelet length. Increasing the filter length to a certain extent leads to stronger subtraction and also increases the time needed to run the procedure.
- **White noise level** – regularization parameter. Increasing this parameter leads to a more stable result and softer subtraction effect.

- **Ghost delay –** static data shift used to generate the ghost wave models
- **Constant delay (ms)** – bulk shift in ms that will be introduced into the data. This value should correspond to the difference between the reflection arrival time and the associated ghost wave arrival time averaged across all traces.
- **From Header** — sets various shifts for each trace from the header field. The user needs to fill in the header field in advance.

- **Use adjacent traces** – inclusion of adjacent ghost wave model traces in filter calculation and averaging of the obtained filters across several traces
- **Number of traces** – this parameter sets the number of adjacent traces of the ghost wave model on each side of the current trace that will be taken into account during the filter generation. If it is equal to zero, adjacent traces will not be used, and the filter will be built based on the single current trace of the model. If it is equal to 1, thee traces will be used: the central one and two adjacent ones on both sides, etc. This parameter is discussed in greater detail in the Wave Field Subtraction module description. Increasing this parameter amplifies the subtraction effect but results in a longer procedure run time. It is a good idea to start with 3 adjacent traces and then try to increase or decrease this number.
- **Filter averaging base (tr)** – sets the number of traces with which the calculated filters will be averaged. Usually it is recommended to enter a relatively large number here. You can start with 25 and work up or down from there. Increasing this parameter results in a softer subtraction, while decreasing it makes subtraction stronger.
- **Band transform** – this option limits the frequency range within which the operations are performed. It allows decreasing the processing time and performing additional frequency filtering of the data.
- The **Low frequency** and **High frequency** parameters set the lower and upper boundaries of the used frequency range, respectively.
- **Output ghost model** – when this option is on, the module does not subtract the model from the original wavefield but output the model itself, instead. This can be used for parameter testing and debugging.

Interpolation

Profile Interpolation* (Interpolation of profile data on a regular grid)

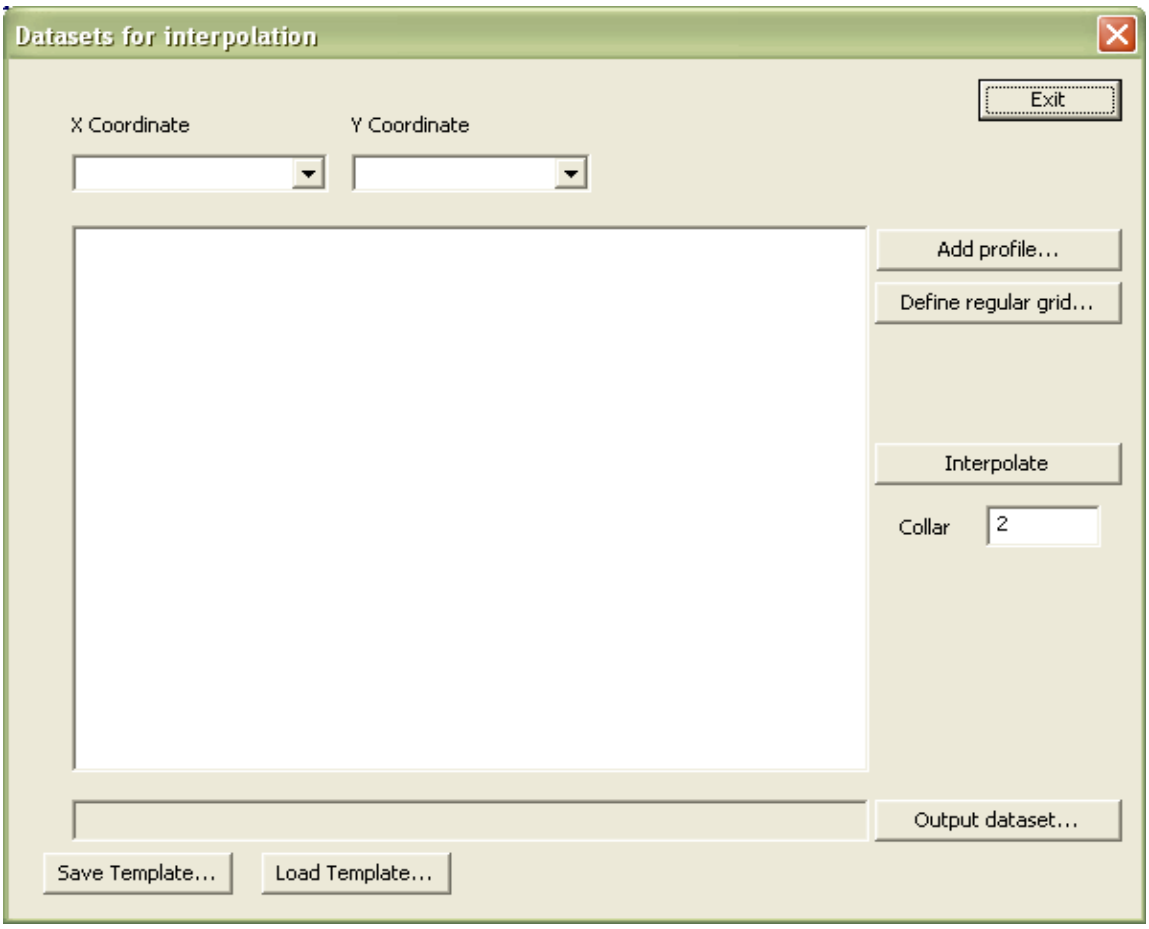
The module is designed for data interpolation into a regular grid. This module belongs to the class of stand alone modules, that's why it doesn't require any input/output modules in the flow.

Input data

The input data for the interpolation procedure are traces that reside in one or several datasets and traces' coordinates, which can reside in any traces' header fields. The requirement for the input data is an equal number of samples.

Module operation

To activate the module you have to put it into the flow and click the button **Run**, after that a parameters' dialogue appears:



Parameters

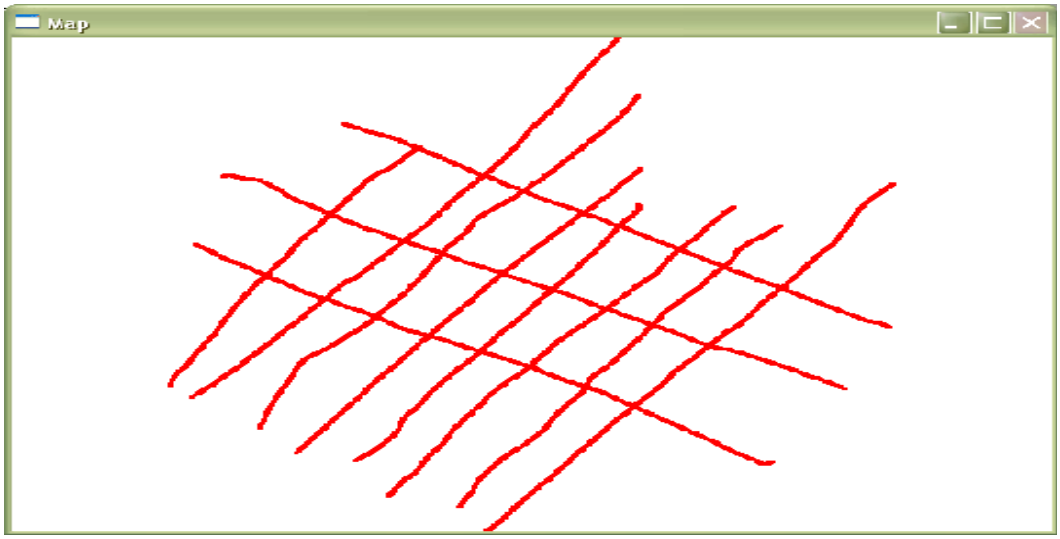
- **X Coordinate** allows selecting x-coordinate for the raw data
- **Y Coordinate** allows selecting y-coordinate for the raw data
- **Add profile...** adds raw data for interpolation, after having added data the name of the dataset appears in the list
- **Output dataset...** allows specifying the dataset, to which the interpolation results will be saved
- **Save Template/Load Template** saves/loads the templates into the database
- **Define regular grid...** produces interactive tool for specifying regular grid
- **Interpolate** triggers the interpolation
- **Collar** specifies the size of vicinity for data retrieval at the edges of the

regular grid, the size is given in steps of a regular grid

- **Exit** exit from the module

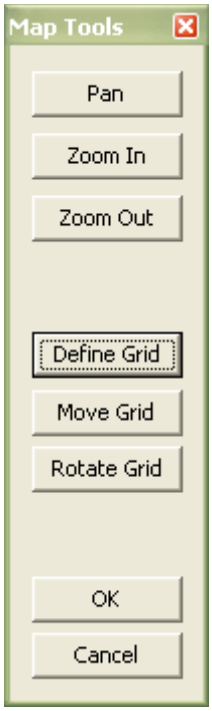
The module operation requires the input data specification (button **Add profile...**) and indication of which trace headers correspond to which x- and y-coordinates (items **X Coordinate** and **Y Coordinate**).

After that you have to specify a regular grid for interpolation, and click the button **Define regular grid...;** as a result a map window appears, where the raw profiles and a regular grid (if any) will be displayed:



as well a window that displays current cursor coordinates appears:

and a control window of display of map and regular grid appears:

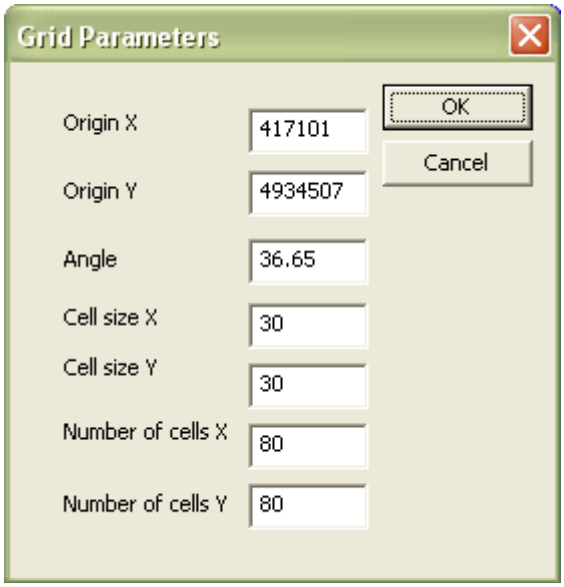


where:

- **Pan** Image positioning of the map
- **Zoom In/Zoom Out** zooming in/zooming out of the display scale
- **Define Grid** defining regular grid parameters
- **Move Grid** moving regular grid
- **Rotate Grid** Rotating regular grid with reference to its' origin
- **OK** Closing dialogue with saving parameters
- **Cancel** Closing dialogue without saving parameters

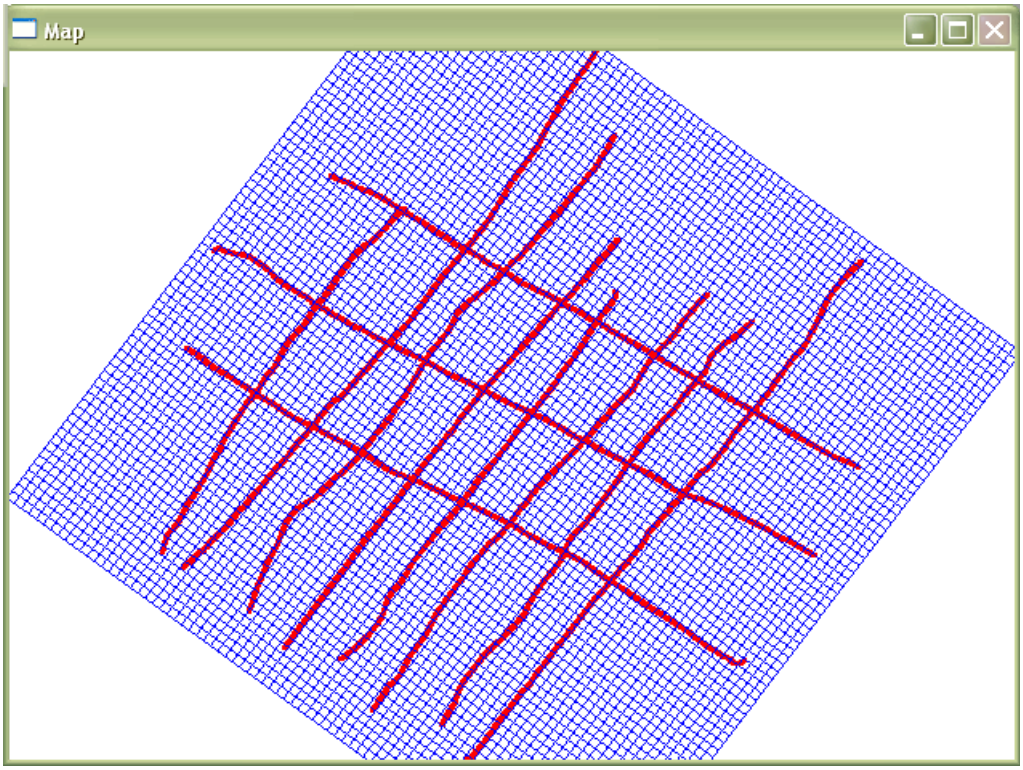
To specify the regular grid you have to click the button **Define Grid**, then parameters' dialogue appears:

where:



- **Origin X/Origin Y** origin of the regular grid
- **Angle** rotation angle of the regular grid
- **Cell size X** cell size of the regular grid in X direction
- **Cell size Y** cell size of the regular grid in Y direction
- **Number of cells X** number of cells of regular grid in X direction
- **Number of cells Y** number of cells of regular grid in Y direction

After specification of the regular grid it will be displayed on the map window:



In case of the necessity the position of the regular grid can be corrected by moving and rotating (buttons **Move Grid** and **Rotate Grid** correspondingly).

Upon completion of the regular grid specification you should click the button **OK** to save the parameters.

Then you have to specify the dataset to save the results of interpolation (button **Output dataset...**) and perform the interpolation (button **Interpolate**).

If required you have to save / load interpolation templates (buttons **Save Template/Load Template**), as after quitting the module the setup is not preserved.

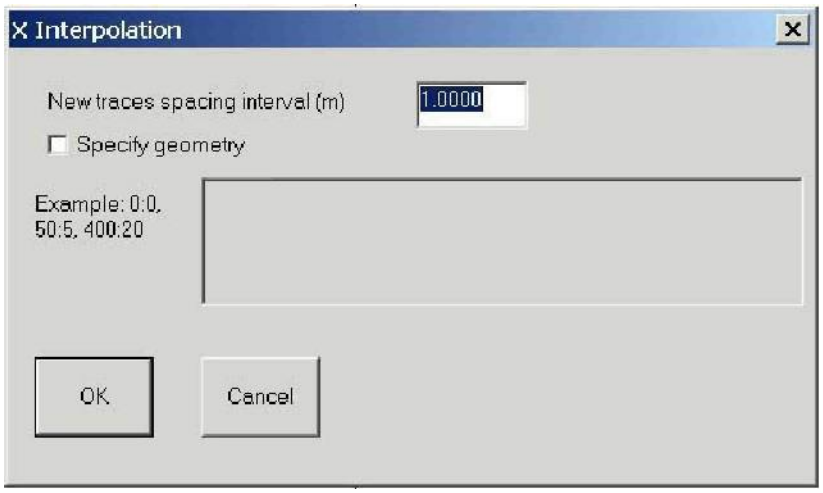
The headers **CDP_X, CDP_Y** in the output dataset will contain coordinates in the system corresponding to the raw data, while the headers **ILINE_NO**, **XLINE_NO** - cell numbers on X- and Y-axis of the regular grid.

CCP-CMP X Interpolation

This module allows accomplishing a dataset interpolation in horizontal direction. The sinc function out of 8 points is used in interpolation algorithm. The dataset under interpolation is re-sampled to a

rarer or more frequent spatial interval. The new step between the traces can not be an integer or a divisible by step values at the input.

When this module is activated the following dialog box opens:



New trace spacing interval - a value that defines the step between the traces at output. This value should be specified in meters.

Specify geometry - in case if in the initial data the step values between the traces were different these values must be reduced to constant by means of interpolation. In this case the **Specify geometry** flag should be marked on and, specify the correspondence between trace numbers and x coordinates (offsets for profile start). First and last traces correspond to coordinates **from** x and **to** x, respectively, specified in the ***Data Input*** module. The correspondence betweenother traces must be assigned in a text field which is activated if the flag is marked on. Geometry specification format looks as the following:

n1:x1,n2:x2,..., nm:xm,

where ni - trace number, xi - coordinates. Trace numbers and coordinates should be specified increasingly. The correspondence between trace numbers and coordinates between adjacent specified n,x pairs is considered as linear.

Refraction

Travel Time Inversion*

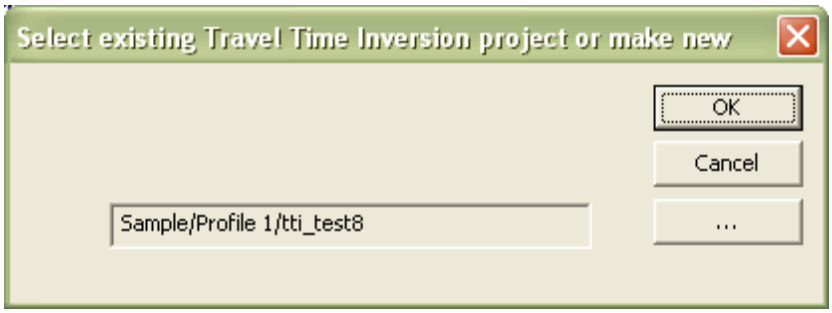
The **Travel Time inversion** module is used to process curved-path refraction first arrival materials using the Herglotz-Wiechert (HW) method as well as the tomographic approach.

Travel Time inversion is a so-called standalone module, i.e. a module that generates the flow by itself.

All data processing within the module takes place within a certain processing *scheme*. A scheme is a combination of first arrival picks, processing and visualization parameters, and the resulting velocity model. Each scheme is stored in a separate directory within the RadExPro project.

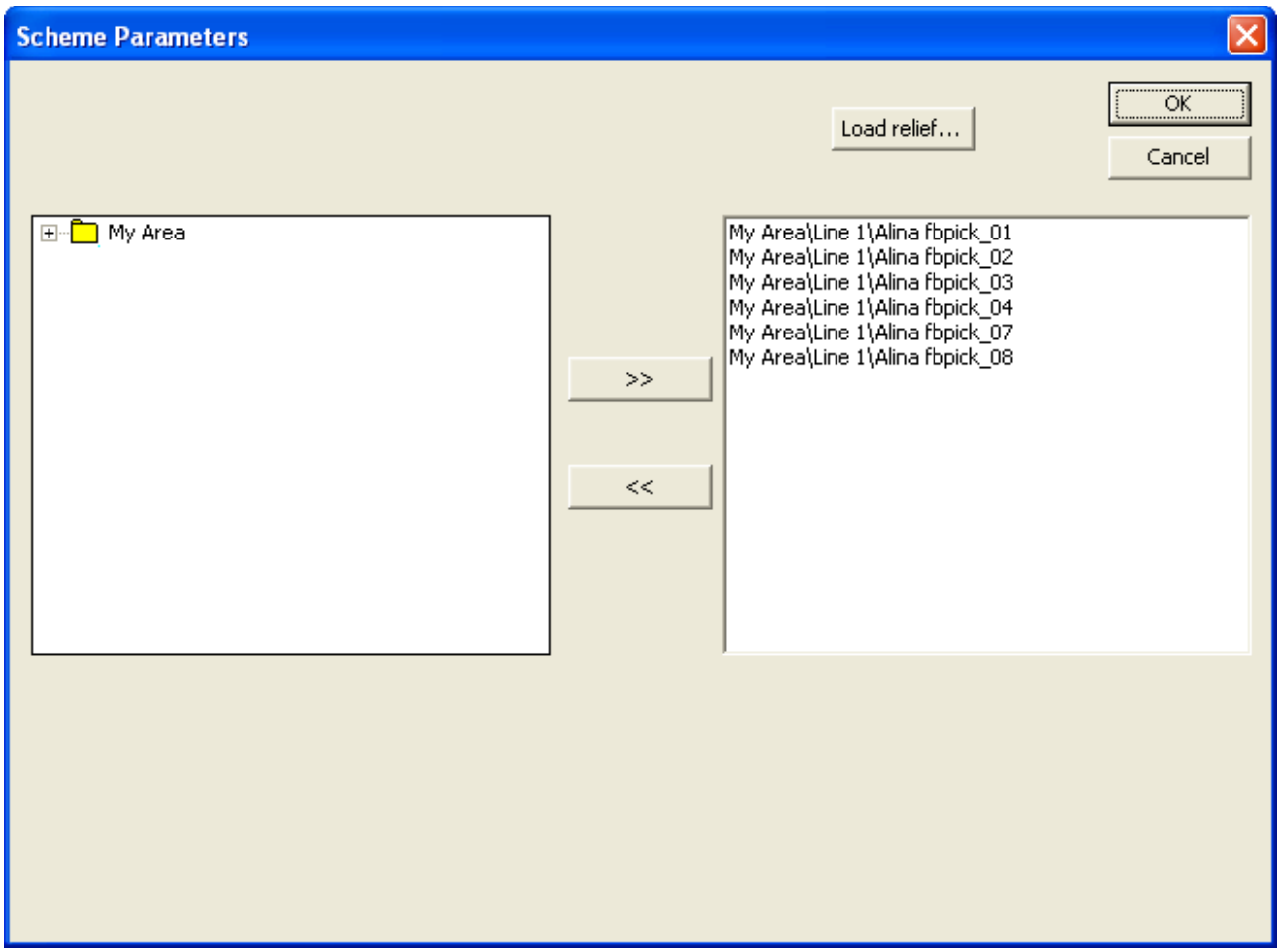
Working with the module

After adding the module to the flow, specify the name of an existing scheme in the **Select existing Travel Time Inversion project or make new** dialog box or create a new one by clicking the **"…"** button.



When execution of the flow is started (**Run)**, the following dialog boxes will appear: **Scheme Parameters** and **Travel Time Inversion.**

Scheme Parameters – this dialog box is used to add first arrival picks to the scheme and loading topography.



The left half of the dialog box contains a viewer means for the project database containing the picks necessary to run the module.

A list of picks added by the user to the scheme is shown in the right half of the dialog box.

Picks are added to the scheme by double-clicking their name in the project database or clicking the >> button.

To delete a pick from the scheme, select it from the list and press **delete** on the keyboard or click the << button in the dialog box.

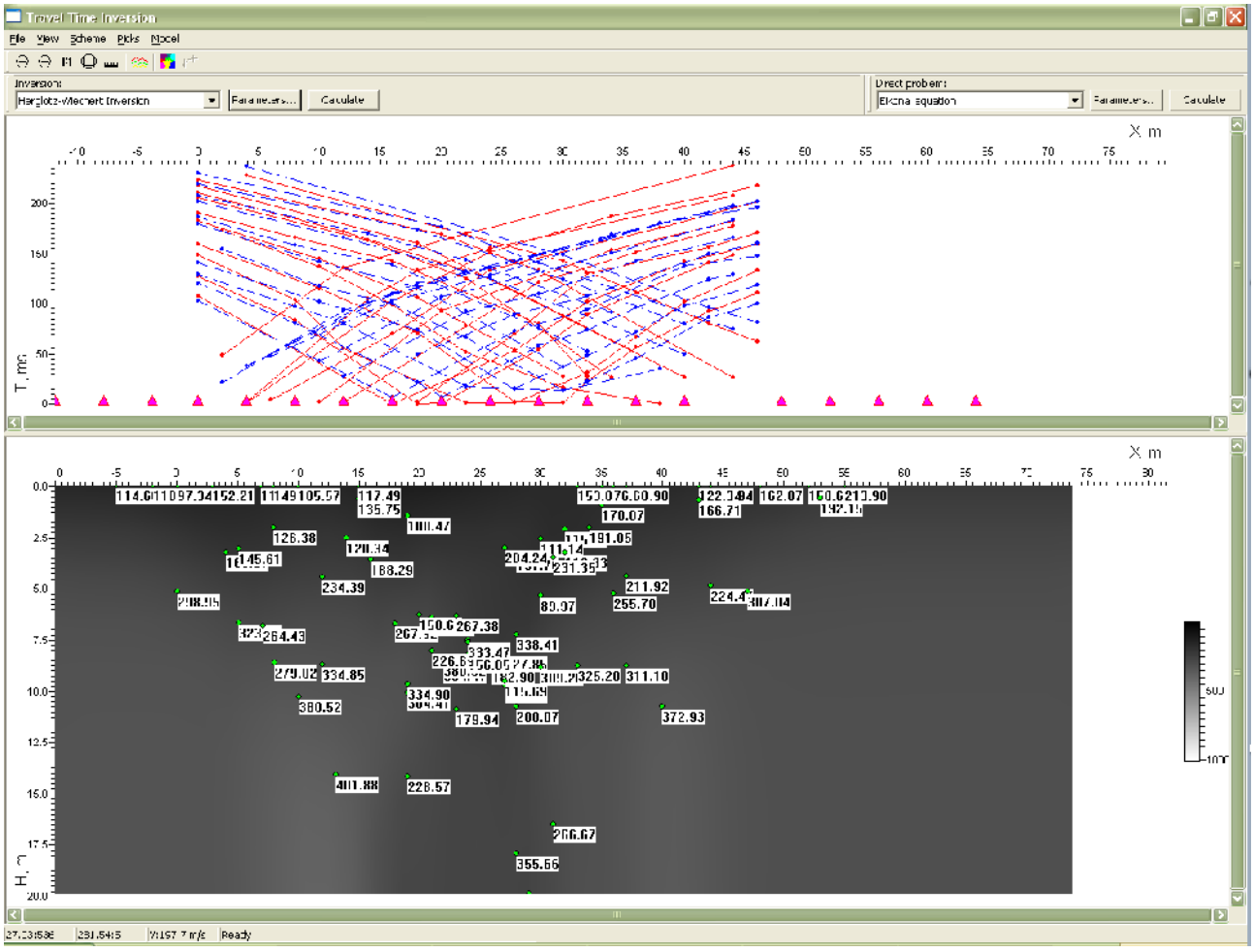
To add a topography line, click the **Load relief**… button and select a relief-defining pick linked to the REC_X header field.

IMPORTANT: For the module to run successfully, the first arrival picks should be linked to the SOU_X and REC_X header fields, and the relief-defining pick should be linked to the REC_X header field. To make sure this is the case, press the **Pick headers** button in the **Save pick** dialog box when saving the pick. A new dialog box with two header lists will open, allowing you to select the necessary fields.

One pick can contain first arrival curves obtained from several shot points (SP). In this case the pick is divided into flank segments when it is imported into the project. Each segment corresponds to one SP and has a unique name.

Travel Time Inversion – the primary working window of the module: the upper part of the window shows the observed travel time curves (in red), theoretical travel time curves (in blue), and source position (in pink); the velocity model, rays, and velocity values for specific points are shown in the lower part of the window.

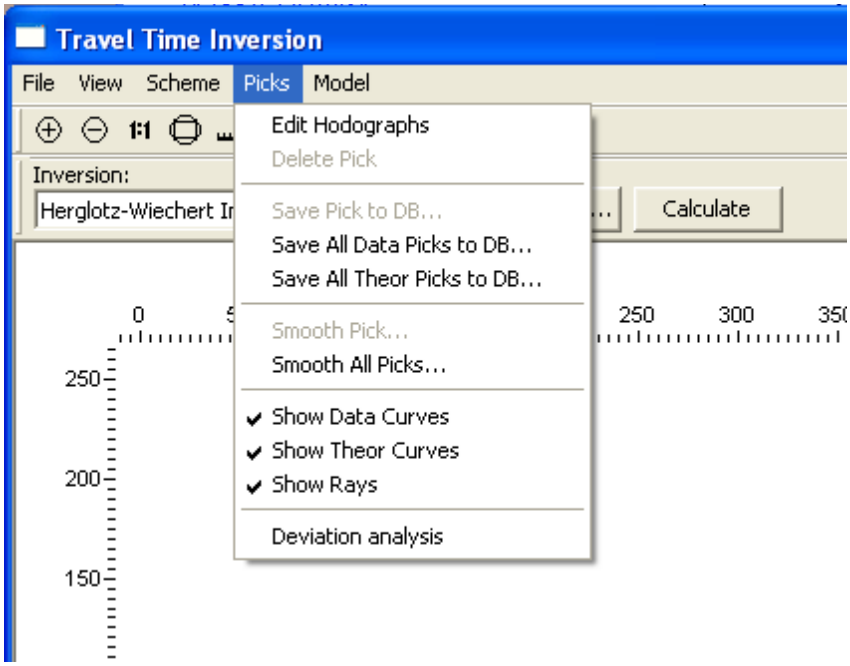
The window also contains a menu bar and a toolbar.



Scheme

This menu opens the **Scheme Parameters** dialog box used to edit the list of picks within the scheme and the topography.

Picks



The Picks menu allows the user to:

- Edit picks (**Edit Hodographs**)
- Delete picks (**Delete Pick**)
- Save an individual pick to the database (**Save Pick to DB**)
- Save all observed picks to the database (**Save All Data Picks to DB**…)
- Save all theoretical picks to the database (**Save All Theor Picks to DB**…)

- Smooth selected pick (**Smooth Pick…**)

- Smooth all picks (**Smooth All Picks**…)

- Enable/disable theoretical picks (**Show Theor Curves**), observed picks (**Show Data Curves**) and rays (**Show Rays**)

- Analyze deviation of the observed picks from the theoretical ones (**Deviation analysis**)

Editing picks

To start editing the picks, select the **Edit Hodographs** menu item or click the button on the toolbar.

The following actions can be performed with picks:

- **Adding a point.** Place the mouse cursor over the spot where a new point is to be created and press the left mouse button (MB1).

- **Moving a point.** Place the mouse cursor over the point to be moved, press and hold the right mouse button (MB2) and drag the point to a new time (hold Ctrl key to move the point horizontally as well)

- **Deleting a point.** Place the mouse cursor over the point to be deleted and double-click the right mouse button (MB2).

- **Deleting several points.** While pressing the Shift key, press and hold the left mouse button (MB1) and select a rectangular area containing all points to be deleted.

- **Selecting a pick.** Move the cursor over the pick, press the Ctrl key and the left mouse button (MB1). Press Tab to cycle through the picks.

- **Moving a pick**. Place the mouse cursor over the pick, press Shift, then press the right mouse button and move the pick while holding down MB2+Shift.

Deleting picks

To delete a pick, first select it and then select **Delete Pick** from the **Picks** menu.

Saving picks

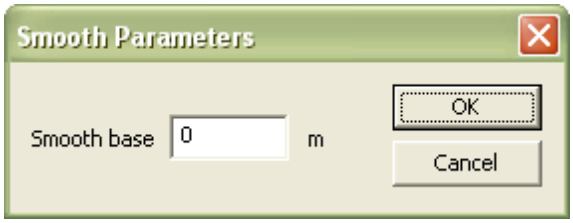
To save a pick, first select it and then select **Save Pick to DB**… from the **Picks** menu. A save dialog box will appear allowing you to specify the pick name and project database section where the pick will be saved.

To save theoretical and observed picks to a separate project database object, select **Save All Theor Picks to DB**… or **Save All Data Picks to DB**… menu item for theoretical and observed picks, respectively. A save dialog box will appear allowing you to specify the pick set name and project database section where the set will be saved.

Pick smoothing

This procedure can be applied only to observed picks (either to an individual pick or to all picks). To perform individual pick smoothing, select it in the editing mode and then select the **Smooth Pick…**menu item or click the button on the toolbar.

The smoothing setup dialog box will open:

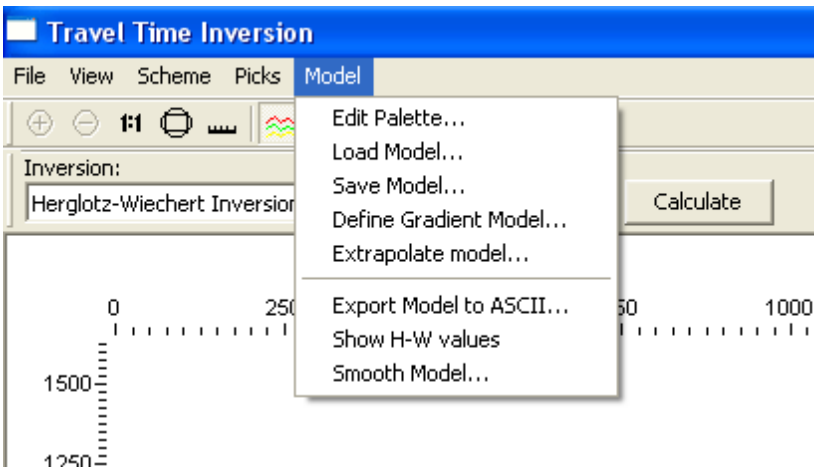


The **Smooth base** parameter is used to specify the averaging window length in meters.

Set this parameter and click **OK**.

To perform smoothing for all observed picks, select the **Smooth All Picks…**menu item and specify the **Smooth base** parameter.

Model

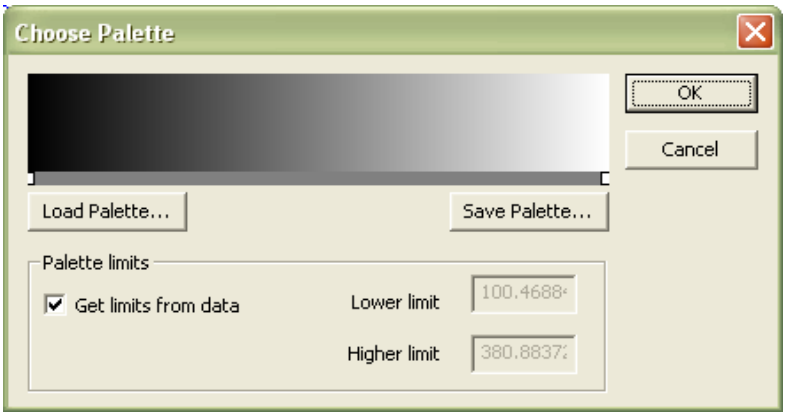


The velocity model menu allows the user to:

- Edit the palette used to display the velocity model (**Edit Palette**…)
- Load a velocity model in the GRD format (**Load Model**…)
- Save a velocity model in the GRD format (**Save Model**…)
- Specify a gradient velocity model manually (**Define Gradient Model**…)
- Extrapolate a velocity model to the edges of the sources and receivers (**Extrapolate Model**…)
- Export a velocity model as a text file (**Export Model to ASCII**…)
- Enable/disable the display of velocity values obtained using the Herglotz-Wiechert (HW) method (**Show H-W values**…)
- Smooth a velocity model (**Smooth Model**…)

Editing the velocity model palette

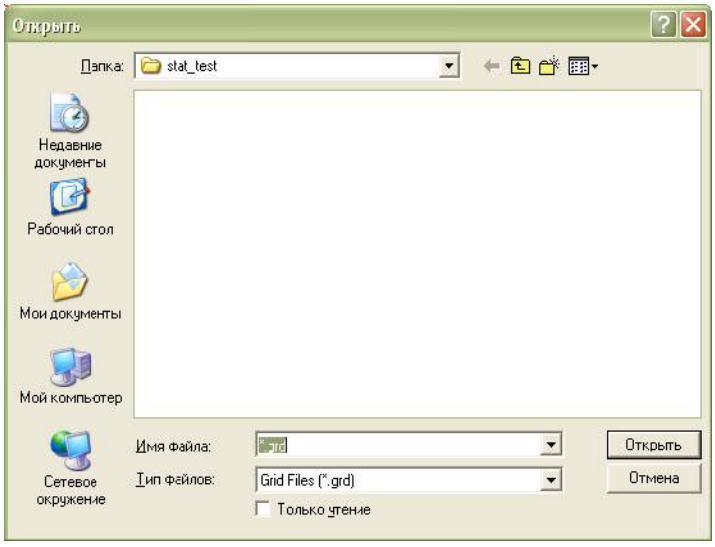
Select the **Model/Edit Palette…**menu option. The palette setup dialog box will open.



Working with palettes in this module is generally similar to working with palettes in the **Screen Display** module and is described in the RadExPro Plus 3.95 User Manual. The only difference is the **Palette limits** field allowing the user to define the velocity value range based on the data (**Get limits from data)** or manually specify the minimum **(Lower limit)** and maximum **(Higher limit)** values.

Loading a velocity model

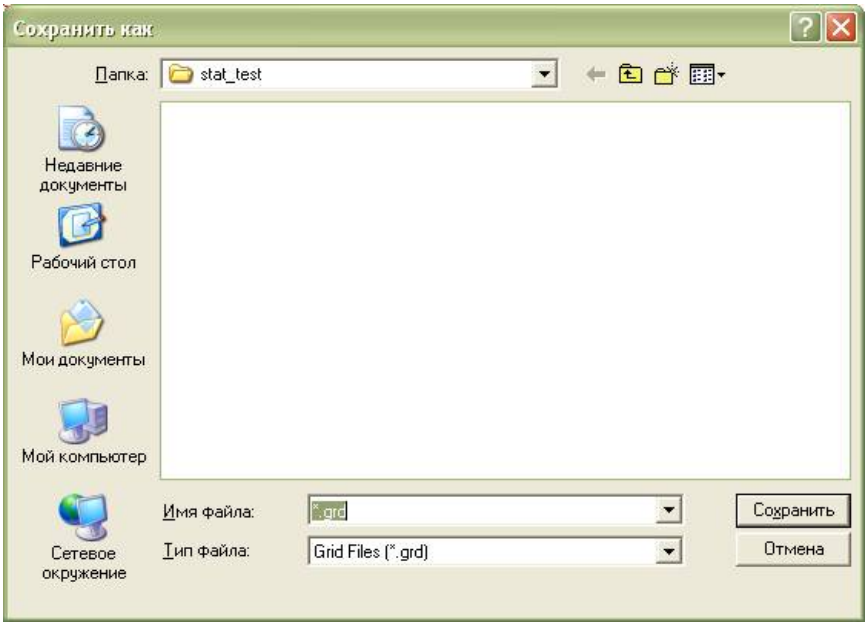
Select the **Model/ Load Model…** menu option. A file selection dialog box will open:



To load a velocity model, select the necessary GRD file in the dialog box.

Saving a velocity model

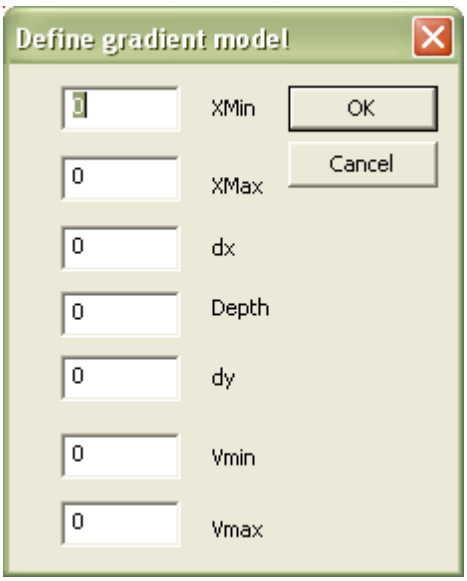
Select the **Model/ Save Model…**menu option. A file selection dialog box will open:



Specify the name of the file to which you want to save the model.

Specifying a gradient velocity model

Select the **Model/ Save Gradient Model…**menu option. The gradient velocity model setup dialog box will open.



XMin: X – velocity model start coordinate

XMax: X – velocity model end coordinate

Dx: velocity model cell width

Depth: velocity model depth

Dy: velocity model cell height

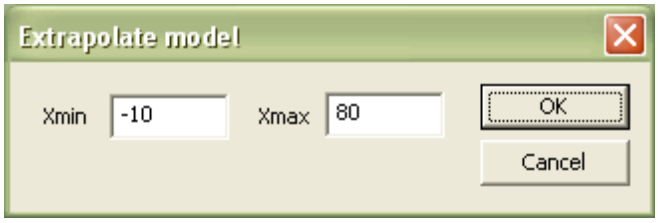
Vmin: velocity model initial velocity

Vmax: velocity model terminal velocity

Set all necessary parameters and click **OK**.

Extrapolating a velocity model

Select the **Model/ Extrapolate Model…** menu option. The velocity model extrapolation setup dialog box will appear.



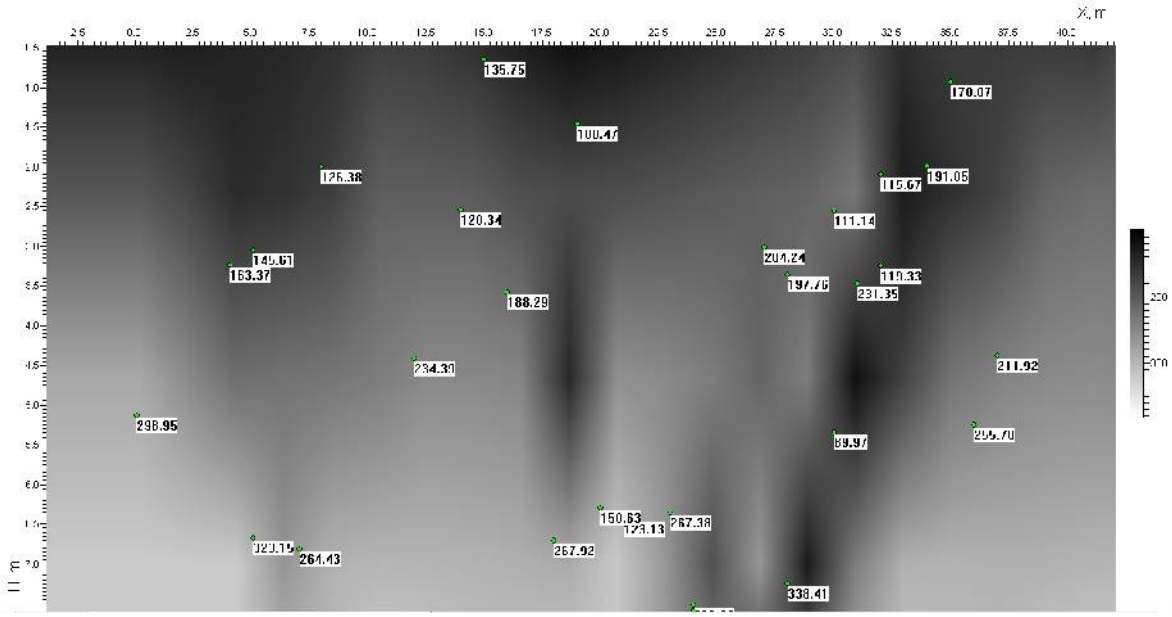
Specify the desired minimum and maximum X coordinates of the model in meters in the **Xmin** and **Xmax** fields.

Exporting a velocity model as a text file

Select the **Model/ Export Model to ASCII…**menu option. In the file save dialog specify the name of the file to which you want to save the model.

Enabling/disabling the display of velocity values obtained using the Herglotz-Wiechert (HW) method

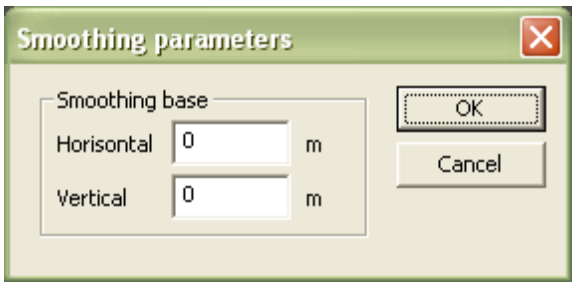
The **Model/ Show H-W values…**menu option enables or disables display of values generated using the HW method (if any).



Velocity model smoothing

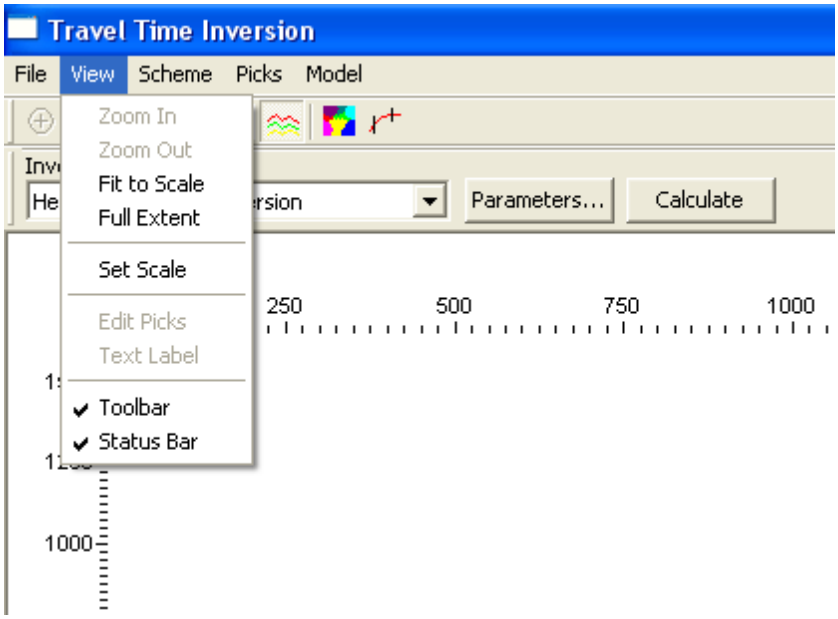
Select the **Model/ Smooth Model…**menu option or press the button on the toolbar.

The velocity model smoothing setup dialog box will open.



Specify the smoothing window **Horizontal** and **Vertical** sizes in meters.

View



The **View** menu option opens a submenu containing the following commands:

Zoom In – increase the current zoom level (this option works similarly to the **Zoom/Set Zoom** menu option of the **Screen Display** module and is described in the RadExPro Plus 3.95 User Manual)

Zoom Out – decrease the current menu level (this option works similarly to the **Zoom/Unzoom** menu option of the **Screen Display** module and is described in the RadExPro Plus 3.95 User Manual)

Fit to Scale – display the image in the "original" size (as set by the **Set Scale** option).

Full Extent – scale the image to fit the window

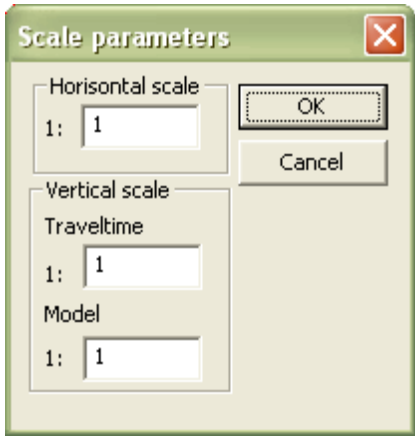
Set Scale – set the scale

Toolbar – enable/disable the toolbar

Status bar – enable/disable the status bar

Setting the scale

Select the **View/ Set Scale** menu option or click the button on the toolbar. The scale setup dialog box will open, allowing the user to specify the horizontal **Horisontal scale** and **Vertical scale** for travel time curves (**Traveltime**) and the velocity model (**Model**).



Inverse problem solution (the Inversion field)

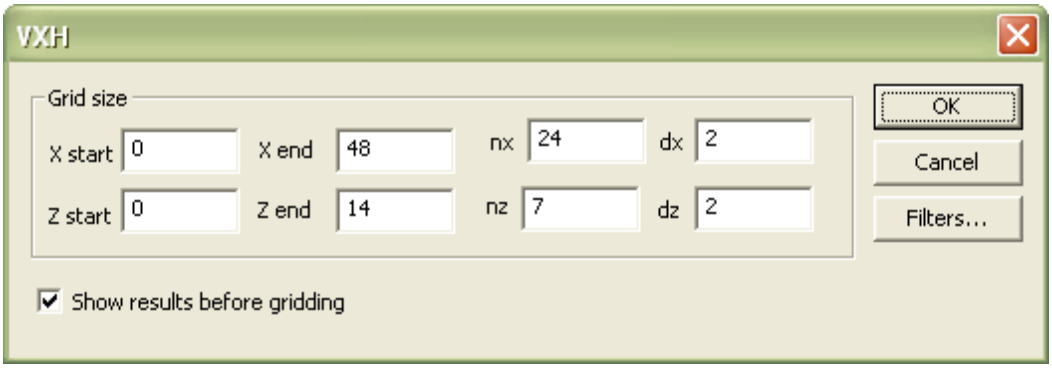
Two methods can be used to solve inverse problems: the HW method (**Gerglotz-Wiechert Inversion**) and the tomographic method (**Tomography**).

HW method (Gerglotz-Wiechert Inversion)

Load the observed travel time curves from the RadExPro Plus database using the **Scheme Parameters** dialog box as described above.

Select **Herglotz-Wiechert Inversion** in the **Inversion** field as the inverse problem solution method**.**

To set the procedure parameters, press the **Parameters**… button. This will open the setup dialog box:



This dialog box allows the user to specify the boundaries of the model (**Grid size**) that will be created when the results obtained using the HW method are interpolated to a regular grid.

X start – model start X coordinate in meters

Z start – model start Z coordinate in meters

X end – model end X coordinate in meters

Z end – model end Z coordinate in meters

nx – number of cells on the X axis in meters

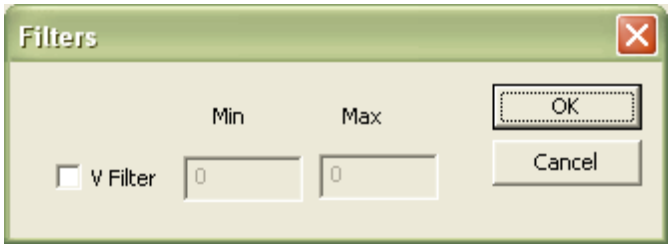
nz – number of cells on the Z axis in meters

dx – cell size on the X axis in meters

dz – cell size on the Z axis in meters

Show results before griding – this parameter enables/disables viewing and editing of results obtained directly using the HW method (a non-regular set of nodes with estimated velocities), before griding them.

Filters… opens a dialog box allowing the user to limit the output data range



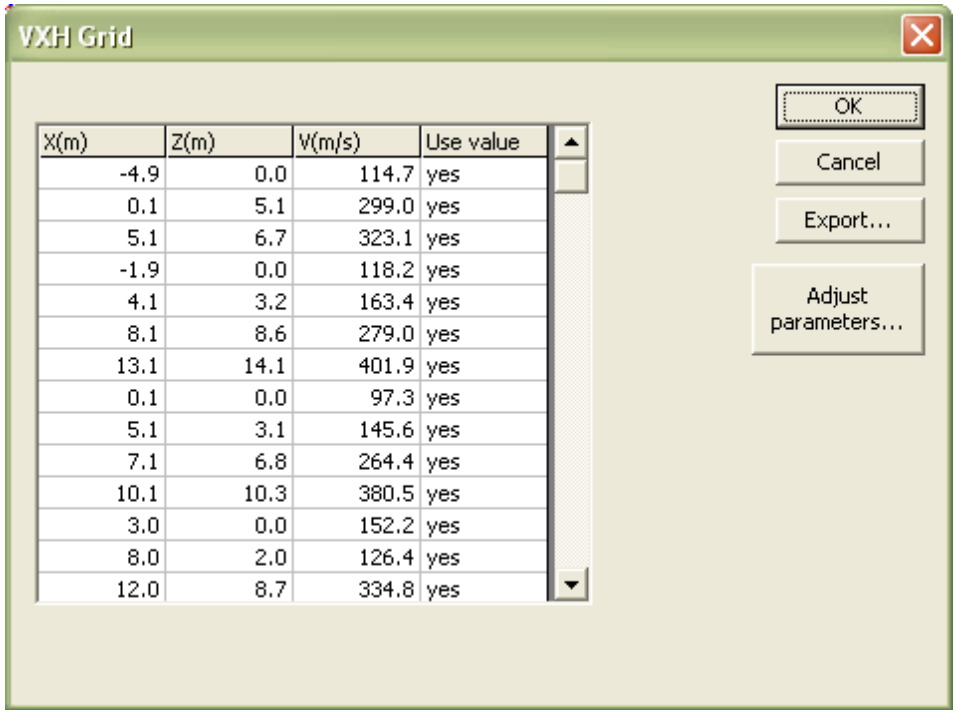
V Filter – this parameter enables/disables the output data filtration mode

Min – minimum velocity value in m/s

Max – maximum velocity value in m/s

If the filter is on, the values outside of the specified range will not be taken for griding.

To run the procedure, click the **Calculate** button. A dialog box will appear (**VXH Grid**):



The calculation results are presented as a table with the first three columns containing X coordinate values in meters, Z coordinate values in meters, and velocity values in meters per second. The

fourth column allows specifying whether the current value will be used in interpolation to a regular grid.

Editing cells

If necessary, the results can be edited. To change the value in a cell, double-click the left mouse button on the cell to be edited or place the cursor over it and press **Enter** key on the keyboard. To exit the editing mode and save the results, press the **Enter** key. If you do not want to save the results of editing, press **Esc key.**

Export… opens a dialog box allowing the user to select an ASCII file name to which the HW calculation results will be exported.

Adjust parameters… opens the interpolation grid setup dialog box (same as the **VXH** setup dialog box).

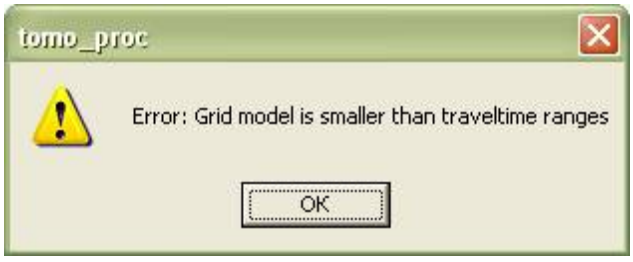
Set all necessary parameters and click the **OK** button. The velocity model built will be displayed built in the lower part of the screen.

To exit the procedure without building a velocity model, press **Cancel**.

Tomography

This procedure requires a source velocity model. It can be obtained using the HW method, specified manually (see **Specifying a gradient velocity mode**) or loaded from an external GRD file. No parameters are needed to run this procedure – just press the **Calculate** button.

If you see the following message,



it means that before running the procedure, you need to extrapolate the velocity model to make its

size along the X axis equal to or larger than the source data receiver range.

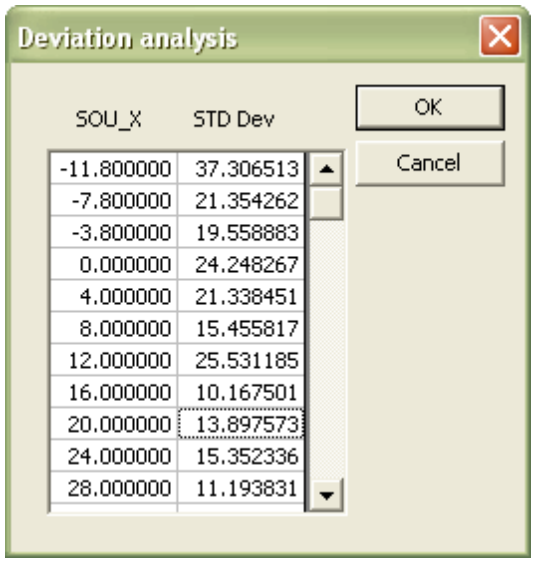
Direct problem solution

Direct problem solution is based on eikonal equation solution and is performed in one of two modes: simple theoretical travel time curve building (the **Eikonal equation** mode**)** and theoretical travel time curve building with ray tracing (the **Raytracing** mode). A velocity model is necessary to solve the direct problem.

Select a mode and click the **Calculate** button.

Analysis of observed travel time curve deviation from theoretical travel time curves (Deviation analysis)

To calculate standard deviation of observed travel time curves from theoretical ones select the **Picks/ Deviation analysis** menu item**.** A dialog box containing a table with two columns will appear: **SOU_X** – shot point coordinate and **STD Dev –** standard travel time curve deviation.



When a table line is selected using the mouse cursor, the travel time curves corresponding to the shot point in the **SOU_X** field in that line will be highlighted in the **Travel Time Inversion** working window.

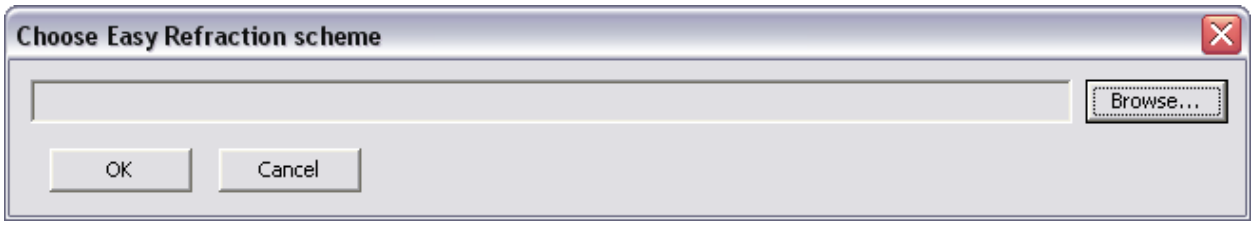
Easy Refraction* (Processing of seismic refraction data)

(This module was developed in cooperation with "Geometria" company)

The module allows processing first arrival curves and building refraction boundaries using the delay-time (classical reciprocal) method.

Easy Refraction is a standalone module, i.e. it shall be the only module in the flow.

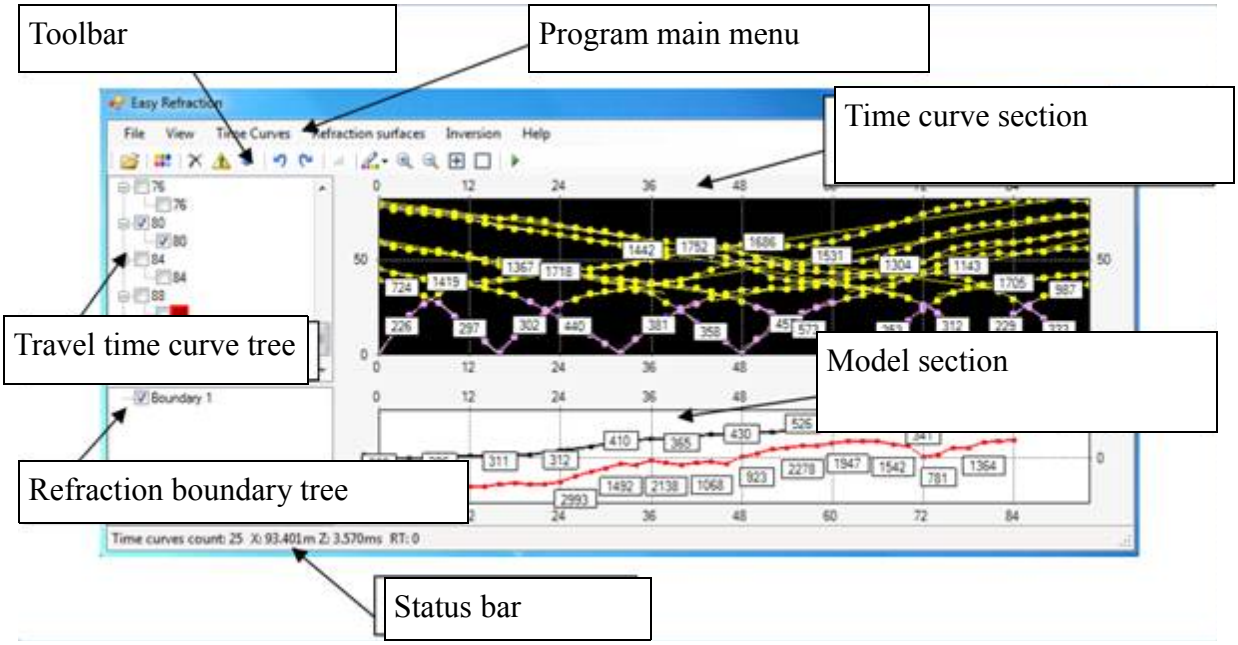
All data processing within the module takes place within a certain processing *scheme*, which is a collection of the time-curves, parameters, and a resulting model. When you add the module into the flow, you will be asked to select a scheme.



Click the Browse button to either select an existing scheme or enter a name of the new one.

Working with the module

The module interface is divided into two main sections – travel time curve section and medium model section:



The travel time curve panel is divided into two parts: the window where the user works with travel

time curves and the tree containing all loaded travel time curves grouped by their source positions.

The model panel is also divided into two parts: the window where the refraction boundaries, day surface relief and wave velocities are shown and the tree containing all refraction boundaries that have been built.

The travel time curve section and model section are synchronized on the X axis in such a way that any scale change or panning in one of the windows results in the same change in another window.

Travel time curves and refraction boundaries are shown as checkboxes in the object tree. If a checkbox is ticked, the corresponding travel time curve/boundary is shown on the screen, if not – it is hidden.

The module allows selecting travel time curves using ether the left or the right mouse button; the selected travel time curves are highlighted in red and blue, respectively. To select, press and hold (!) the corresponding mouse button. If travel time curves have reciprocal points, the reciprocal point time determined by the red travel time curve is shown as two crosshairs on the screen.

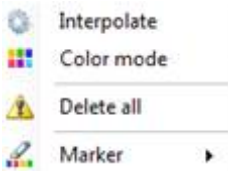
The travel time curve section is divided into two parts: the window where travel time curves are shown and the tree containing all loaded travel time curves associated with their source positions.

The model section contains a tree of all refraction boundaries that have been built.

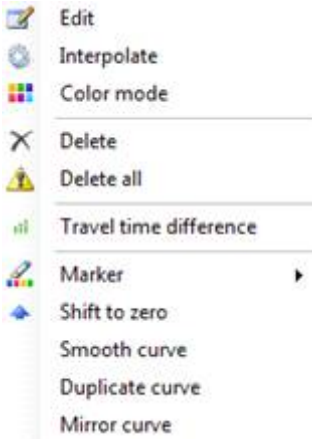
The module allows selecting travel time curves using either the left or the right mouse button; the selected travel time curves are highlighted in red and blue, respectively. If two travel time curves are selected and they have reciprocal points, the reciprocal point time determined by the red travel time curve is shown as two crosshairs on the screen. The status bar at the bottom also shows the value of the time difference at the reciprocal points.

The status bar in the main window of the Easy Refraction module shows the total number of the loaded travel time curves, the X and Y coordinates of the current cursor position, and the difference between the reciprocal times of the two selected travel time curves (if the selected travel time curves have any reciprocal points).

The Easy Refraction module has two different types of context menus opened by single-clicking the right mouse button in the travel time curve editing window. The first type appears if no travel time curve (red) was previously selected with the left mouse button:



If a travel time curve was selected before opening the context menu, the latter will look as follows:



The context menu items are the same as the ones in the ***Time Curves*** menu accessed through the main menu of the Easy Refraction module and will be described in detail further in this document.

Hotkeys

Hotkey:	**Action:**
a	Make selected curve visible/invisible
Ctrl + a	Make all curves visible/invisible
c	Enable/disable color mode
t	Show/hide velocity values by layers
v (press and hold)	Determine velocity using the “ruler” tool
w	Show/hide velocity values by layers and velocity lines
z	Tie the travel time curve to zero
s	Smooth the selected travel time curve
Delete	Delete the selected curve
LMB Double Click	Edit the selected travel time curve
RMB Click	Open the context menu in the travel time curve editing window
Mouse Wheel	Zoom
Ctrl + Mouse Wheel	Vertical zoom
Alt + Mouse Wheel	Horizontal zoom
Space + Mouse Move	Rectangle zoom
MMB Double Click	Return to the full-screen mode
RMB Double Click	Return to the full-screen mode
Ctrl + MMB Double Click	Return to the full-screen mode by the vertical axis
Alt + MMB Double Click	Return to the full-screen mode by the horizontal axis
MMB + Move	Panning
Up, Left, Right, Down	Panning

Shift + LMB + Move	Move the selected travel time curve
Shift + RMB + Move	Move the point on the selected travel time curve
1, 2, …, 9	Enable marker for layer 1, 2, …, 9 or disable it if already enabled
~	Disable marker mode
0	Reset layer marking for all travel time curves
Ctrl + 1, 2, …, 9	Show/hide layer 1, 2, …, 9
Ctrl + 0	Show/hide travel time sections that are not associated with any layer
Ctrl + z	Undo the last action
Ctrl + y	Redo the last action
F5	Launch automatic inversion

Mouse Wheel – mouse wheel rotation

Mouse Move – mouse movement

LMB, MMB, RMB – left, middle, and right mouse button, respectively

Up, Left, Right, Down – up, left, right, and down arrows

Menu items

File

The **File** menu item is used to import/export data (travel time curves, boundaries, relief, velocities) to/from the Easy Refraction module.

The module allows exchanging data both inside the RadExPro project and with external sources.

The module allows saving/loading the current project that contains all information on the travel time curves, boundaries, relief and velocities obtained in the course of working with the module.

The module also allows exporting:

– refraction boundaries in the DWG format,

– travel time curves in the ST format (for further processing in the Zond2DST seismic tomography program).

Load from RadExPro DB

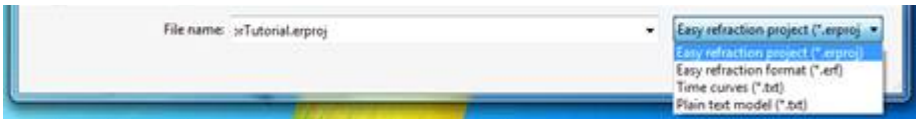
Clicking the **Load from RadExPro DB** menu item opens the following dialog box:

All current project flows are shown on the left; the picks selected for import into the Easy Refraction module are shown on the right. To import picks saved in the project into the Easy Refraction module, select them in the left part of the window and press the >> button.

First arrival curves must be saved with correctly completed SOU_X and REC_X headers to enable their processing in the Easy Refraction module.

Import

Clicking the **Import** menu item opens a dialog box that allows importing data from external files (outside the RadExPro project):



The following input data types can be selected in this dialog box:

- **Easy refraction project**

Easy Refraction module project with the *.erproj extension. This format contains all information on travel time curves, boundaries, relief and velocities obtained in the course of working with the module.

- **Easy refraction format**

File with the *.erf extension. Data are stored in a binary format. This file type is created by the Easy Refraction module for the purposes of data exchange between projects. The file contains travel time curves with selected sections associated with different refraction boundaries in the Easy Refraction module format.

- **Time curves**

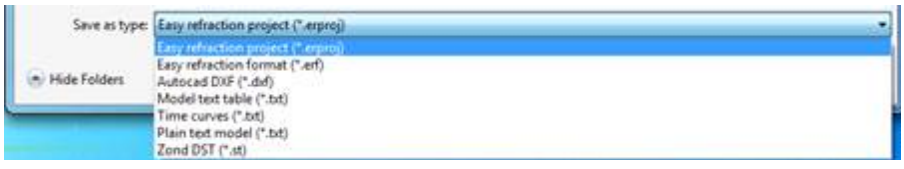
File in the ASCII format consisting of three columns separated by spaces, colons or tabs. The first column is the X coordinate of the source, the second one is the X coordinate of the receiver, and the third one is the time.

- **Plain text model**

File in the ASCII format containing information on refraction boundary positions, relief and velocities as plain text.

Export

This menu item is used to export data for further processing in other applications.



- **Easy refraction project**

Easy Refraction module project with the *.erproj extension. This format contains all information on travel time curves, boundaries, relief and velocities obtained in the course of working with the module.

- **Easy refraction format**

File with the *.erf extension. Data are stored in a binary format. The file contains first arrival curves with selected sections associated with different refraction boundaries in the Easy Refraction module format.

- **Autocad DXF**

File in the DXF format containing refraction boundary positions and a relief line for further formatting of the results in AutoCAD.

- **Model text table**

File in the ASCII format containing text information on refraction boundary positions, relief and velocities in the form of a table.

- **Time curves**

Files in the ASCII format containing three columns – source coordinates, receiver coordinates and first arrival times.

- **Plain text model**

File in the ASCII format containing information on refraction boundary positions, relief and velocities as plain text.

- **Zond2DST**

File in the ST format serving as an input file for the Zond2DST seismic tomography program.

- **Clear project**

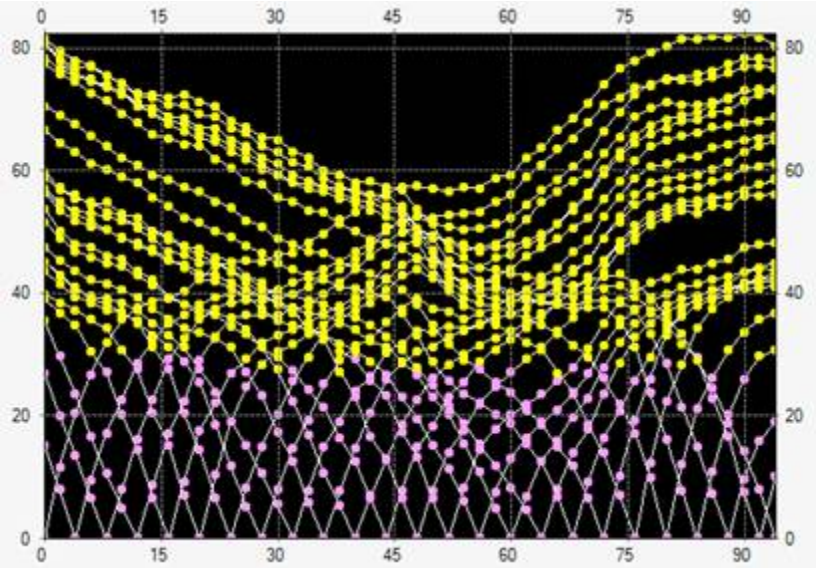
This menu item deletes all data from the current project.

View

The **View** menu item is used to hide/show the travel time curve and model panels and select the travel time curve display color scheme.

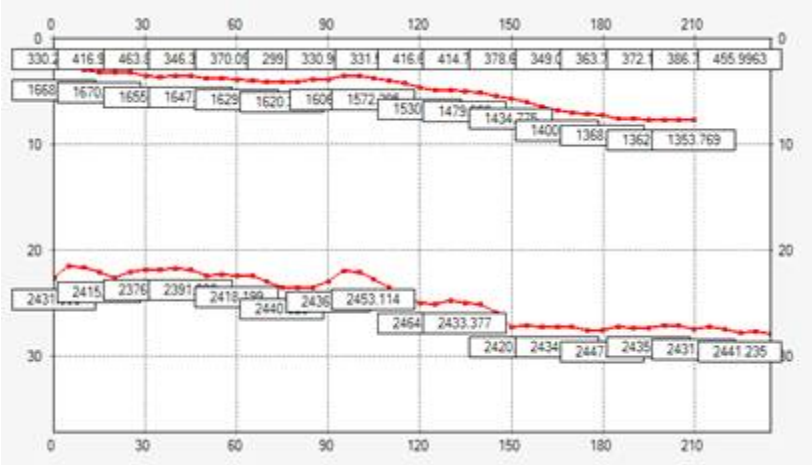
Time curve section

This menu item is used to hide/show the travel time curve section:



Model section

This menu item is used to hide/show the medium model section:



Color settings

Clicking this menu item opens a dialog box that allows setting up the color scheme for the travel time curve section:

- **Plot background color**

Background color for the travel time curve section (black by default).

- **Default curve color**

Travel time curve display color (white by default).

- **Marker Colors**

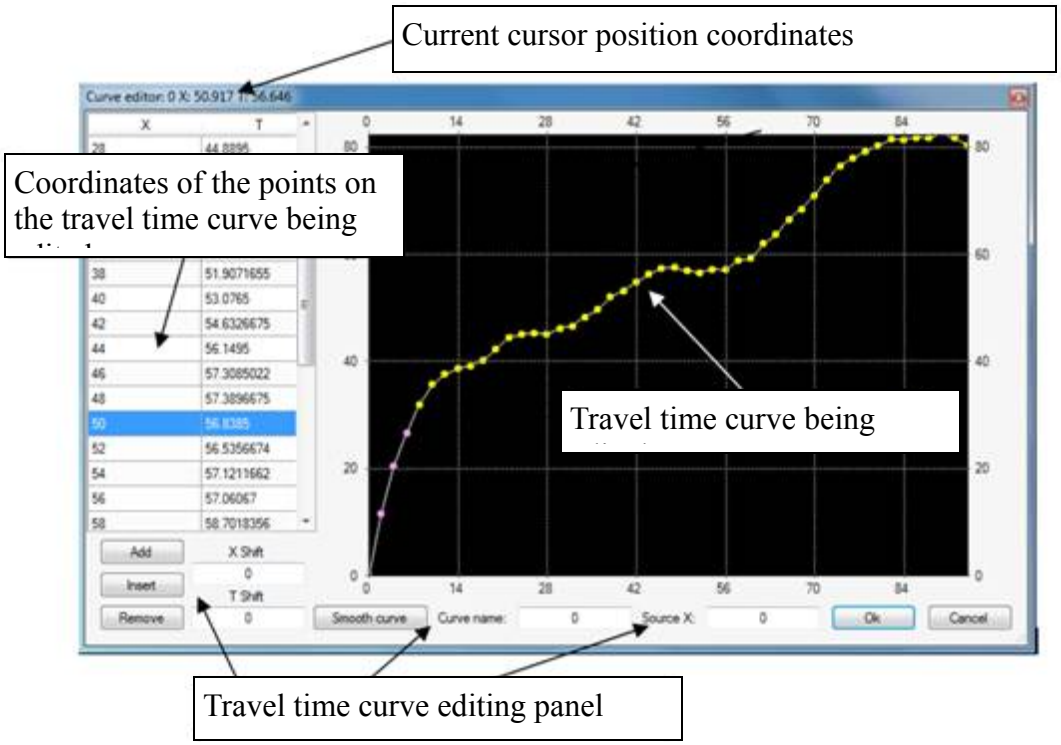
Colors used to display travel time curve sections associated with different refraction boundaries (1 through 9).

Time curves

The **Time curves** menu item is used to edit travel time curves.

Edit

Select this menu item to edit the travel time curve. Clicking it opens the editing window for the selected travel time curve:


Current cursor position coordinates
Coordinates of the points on the travel time curve being
Travel time curve being
Travel time curve editing panel

The travel time curve editing window can be divided into three main sections:

– travel time curve visualization section,

– table with travel time curve point coordinates,

– travel time curve editing panel.

The travel time curve visualization section dynamically displays changes applied to the travel time curve. In terms of functionality it is the same as the travel time curve section of the Easy Refraction module main window.

The table shows the coordinates of the points on the travel time curve. These coordinates can be changed manually, resulting in the travel time curve being redrawn.

The travel time curve editing panel contains the following buttons and fields:

- **Add**

Adds a point to the travel time curve (at the end of the coordinate table).

- **Insert**

Inserts a point into the travel time curve (the point is added between the selected line in the coordinate table and the next line).

- **Remove**

Deletes the selected point on the travel time curve.

- **X Shift**

Shifts the travel time curve by a constant value along the horizontal axis (coordinate axis).

- **T Shift**

Shifts the travel time curve by a constant value along the vertical axis (time axis).

- **Smooth curve**

Smoothes the travel time curve with a floating window by three points.

- **Curve name**

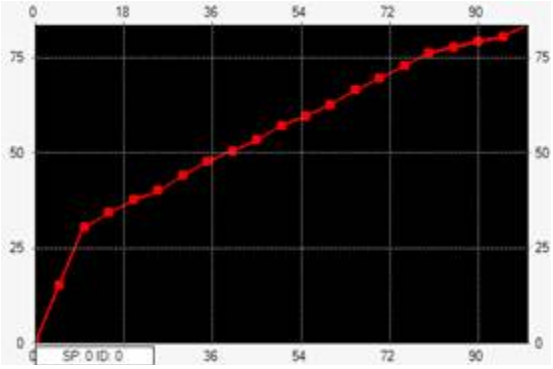
Allows changing the name of the current travel time curve.

- **Source X**

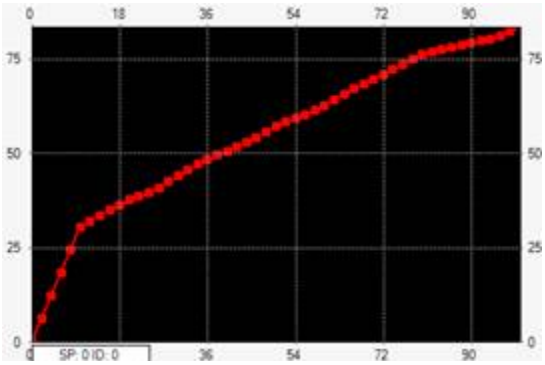
Allows changing the travel time curve source position coordinates.

Interpolate

This menu item is used to interpolate the entire travel time curve system to a new interval by receiver positions.

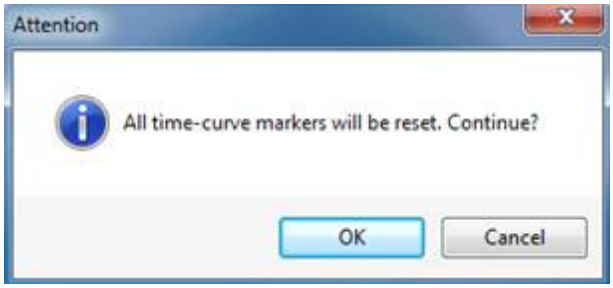


Travel time curve with a 5 m excitation interval.

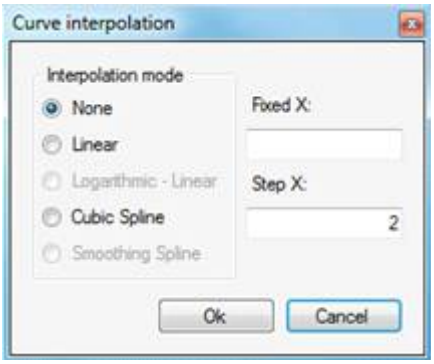


Travel time curve with a 2 m excitation interval.

When this menu item is selected, a dialog box pops up, warning the user that all layer markings will be reset:



Then the interpolation dialog box opens:



- **None**

No interpolation.

- **Linear**

Linear interpolation.

- **Cubic Spline**

Cubic spline interpolation.

- **Fixed X**

Value of the X coordinate from which the interpolation will start.

- **Step X**

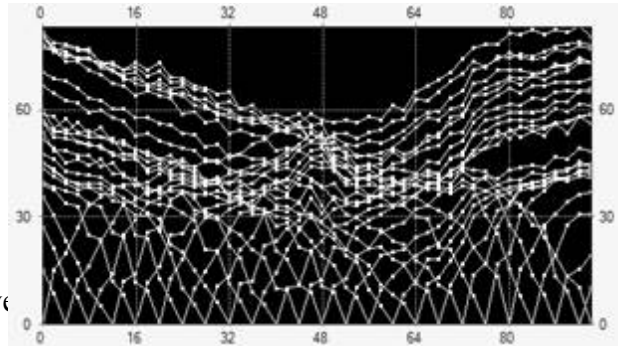
Target source point interval for interpolation.

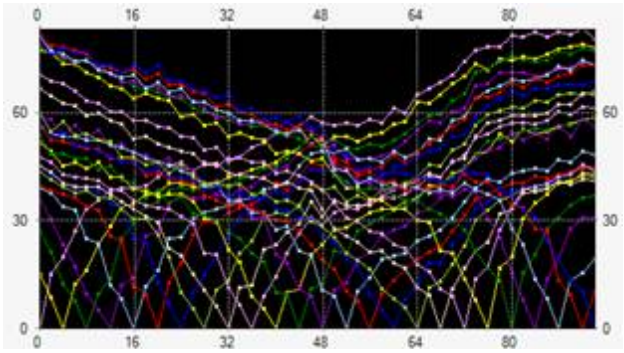
Color mode

When this mode is active, each travel time curve in the list is displayed with a different color.

Standard travel time curve display

Travel time curve





Delete

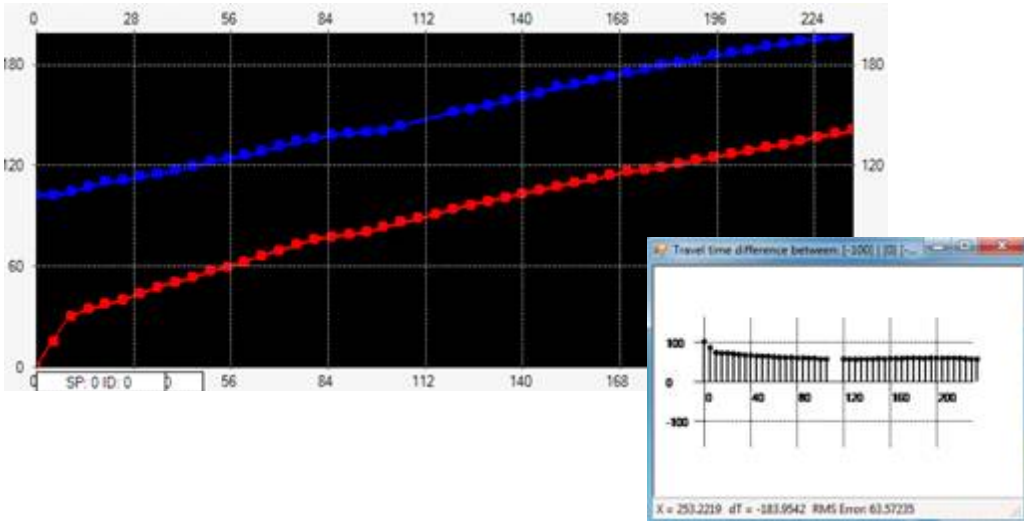
This menu item deletes the selected travel time curve.

Delete all

This menu item deletes all loaded travel time curves.

Travel time difference

This menu item allows building the difference between the two selected travel time curves to evaluate refraction in the medium.



Shift to zero

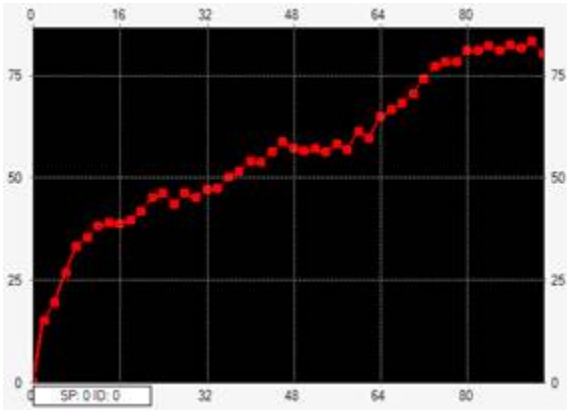
This menu item adjusts all first arrival time values so that the first arrival time on the source coordinate becomes zero.

Marker

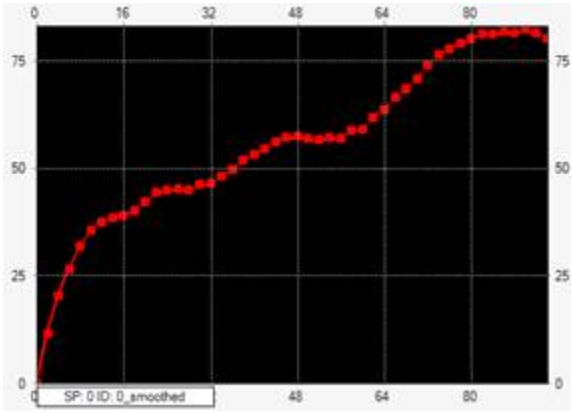
Highlights travel time curve parts associated with different refraction boundaries. Up to 9 refraction boundaries can be built in the Easy Refraction module.

Smooth curve

Smoothes the travel time curve with a floating window by three points.

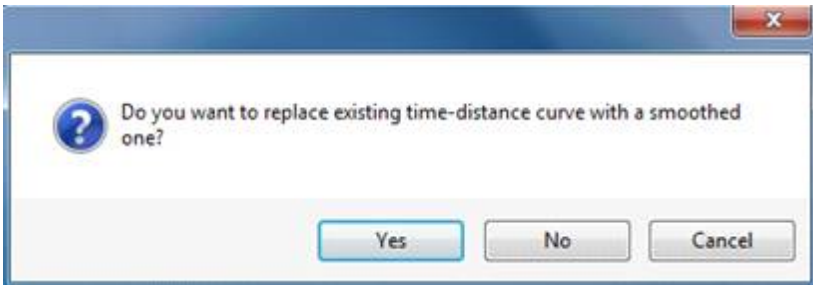


Initial travel time curve



Smoothed travel time curve

Selecting this menu item opens a dialog box prompting the user to save or overwrite the source travel time curve:

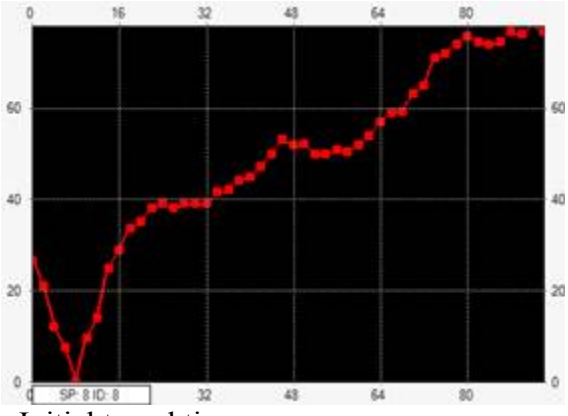


Duplicate curve

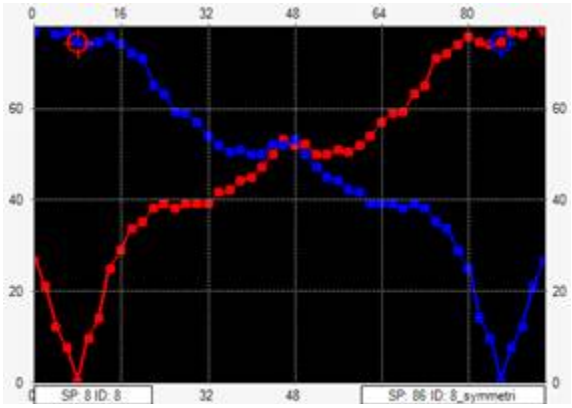
This menu item creates a copy of the selected travel time curve.

Mirror curve

This menu item creates a mirror copy of the selected travel time curve.



Initial travel time curve



Initial and mirrored travel time curves

Refraction surfaces

Delete all

This menu item deletes all refraction boundaries.

Relief

This menu item allows working with the topography.

- **Import**

Imports the relief line from an external ASCII file consisting of two columns separated with spaces, colons or tabs. The first column is the X coordinate and the second one is the absolute relief elevation.

- **Export**

Exports the relief line to an external ASCII file.

- **Clear**

Deletes the relief line.

Keep aspect ratio

If the **Keep aspect ratio** mode is selected, 1:1 scale is maintained for the X coordinate and the depth when performing scaling in the model section.

Inversion

This section contains inverse problem solving methods for the seismic refraction method.

Reciprocal method

The delay-time (classical reciprocal) method for solving refracted wave inverse problems.

Automatic inversion

Automatic inverse problem solving using the delay-time method. When automatic mode is selected,

the processing sequence is as follows:

– building of composite travel time curves,

– leveling of composite travel time curves by the reciprocal time,

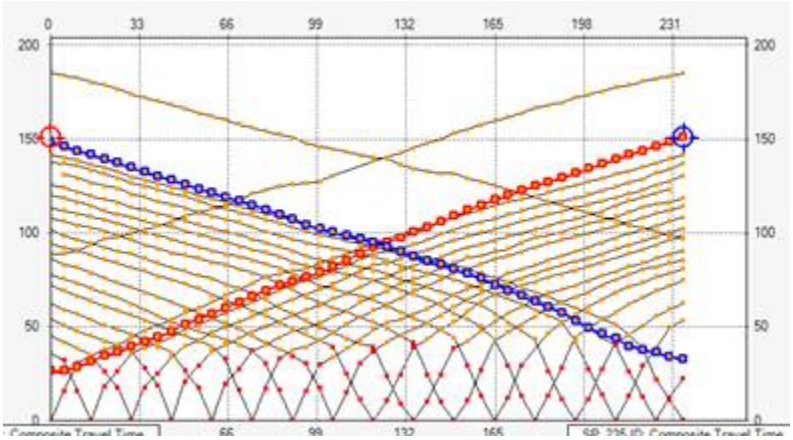
– building of a residual travel time curve and a t0 curve,

– building of refraction boundaries.

If refraction boundaries cannot be built automatically due to the survey system complexity, they can be created manually by following all the procedures step-by-step.

Composite travel times curves

Composite travel time curve building procedure. Composite travel time curves are built based on the sections of the same color (associated with the same refraction boundary):

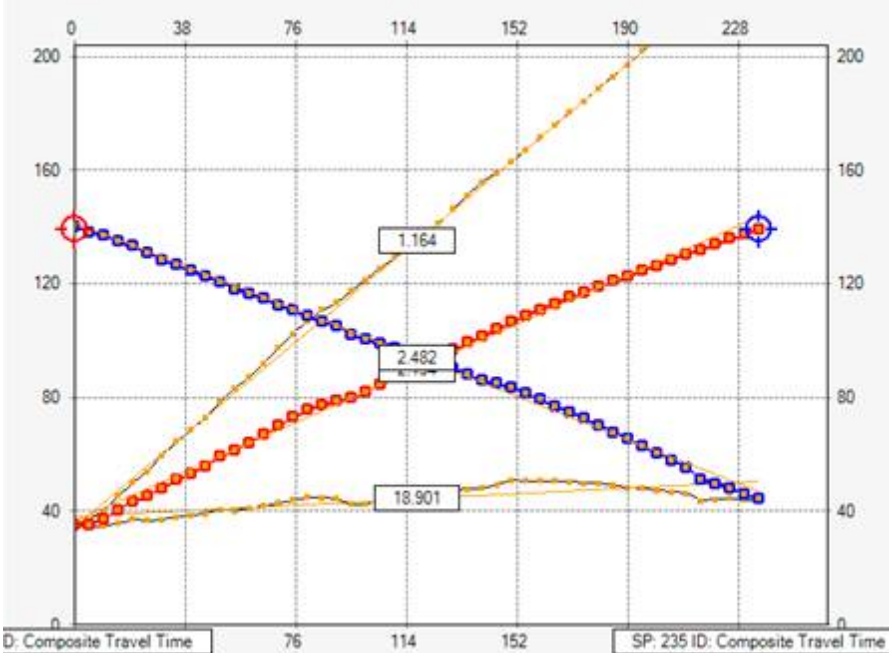


Reciprocal time leveling

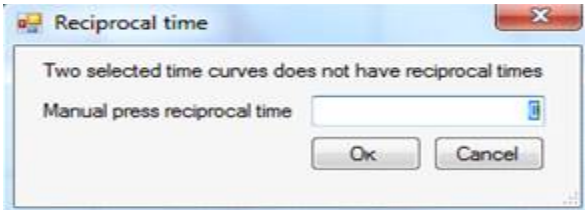
This menu item levels two selected travel time curves by the reciprocal time. If travel time curves have no reciprocal points, leveling is not performed.

Velocity analysis and time-depths functions

This menu item builds a residual travel time curve and a t0 curve for further analysis of the velocity under the refracting surface and building of refraction boundaries. Travel time curves are created automatically based on two selected (!) composite travel time curves.

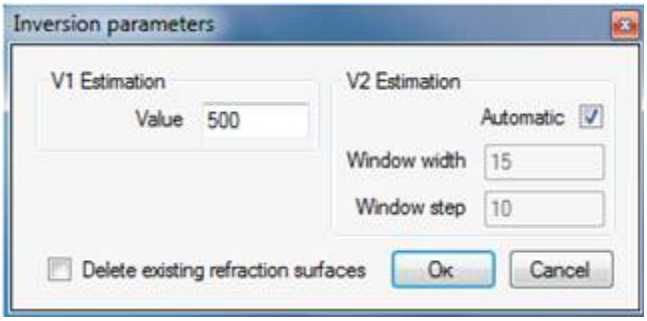


If the two selected composite travel time curves have no reciprocal points, the reciprocal time for calculation of the residual travel time curve and the t0 travel time curve is entered manually:



Refraction surfaces

Refraction boundary building procedure. To build a single refraction boundary, select the t0 travel time curve by left-clicking it and the residual travel time curve by right-clicking it. Enter the value of the velocity in the overlying stratum in the dialog box.



The velocity below the boundary can be calculated using automatic selection of the interval for determination of the velocity by the residual travel time curve and the averaging base (**Automatic** mode). Alternatively, these parameters can be entered manually.

Surface Wave Analysis

MASW (Multichannel Analysis of Surface Waves)

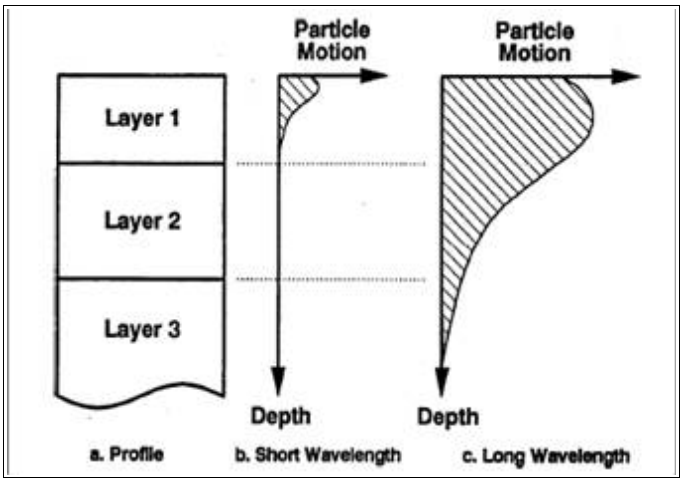
This module is used to create models of S-wave velocities in the near-surface section using surface wave analysis.

This manual is structured as follows: 1) summary of the underlying theory with description of the survey method, 2) general procedure of working with the module – this section describes the procedure of obtaining a S-wave velocity section, from data loading to generation of the final model, 3) detailed description of module functions.

Summary of the underlying theory

During surface excitation of elastic vibrations, over 2/3 of the energy is expended on generation of low-velocity Rayleigh surface waves. In case of a vertical velocity gradient, each frequency component of the surface wave propagates at its own phase velocity, i.e. the surface wave velocity is a function of the vibration frequency.

The surface wave vibration propagation depth is proportional to the wavelength (or inversely proportional to the frequency). On the figure shown below there is a graphic representation of this statement: the high-frequency wave quickly dies out and characterizes the first layer, while the low-frequency wave propagates deeper and provides a characteristic of the lower layers (Rix G.J. (1988))



Geometric dispersion in layered media (Rix G.J. (1998))

Dependence of the phase velocity on the frequency for the particular medium is called a dispersion

curve. The shape of the dispersion curve reflects the change of rigidity with depth.

The above property of dispersion is most commonly utilized to build S-velocity profiles using multi-channel surface wave analysis.

The procedure of working under this method is as follows:

1) **Registration of surface waves** generated by various sources – such as a sledgehammer. Since surface wave energy is considerable, a high signal/noise ratio is easily achieved (in this case body waves act as noise; the area in which they are most active is called the near field).

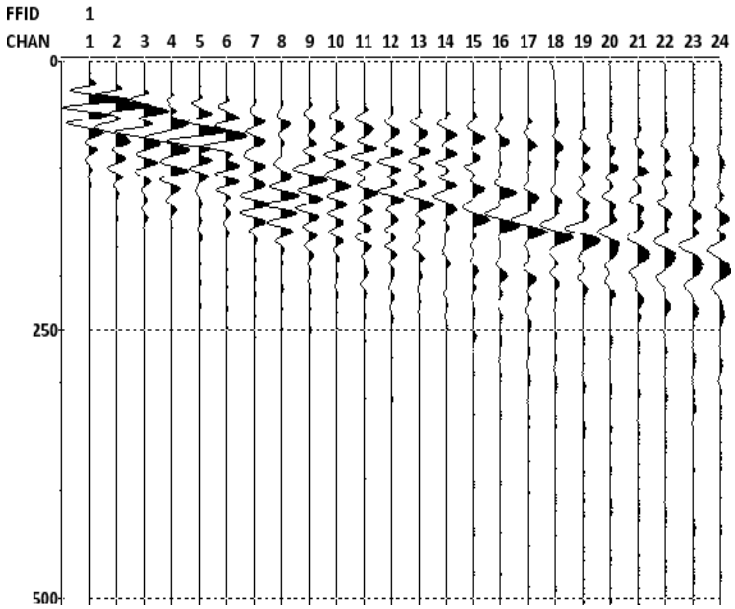
It is recommended to use low-frequency (4.5 Hz) vertical receivers. Use of low-frequency receivers allows registering waves with larger wavelength, which increased the method application depth. Higher frequency receivers may also be used. The length of the receiver line (D) is related to the maximum wavelength (λmax), which, in turn, determines the maximum survey depth: $D \approx \lambda max$, survey depth in this case is defines as a half of wavelength: $Zmax \approx \lambda max/2$.

On the other hand, the distance between the receivers (dx) is related to the minimum wavelength (λmin) and, therefore, the minimum survey depth (Zmin): $dx \approx \lambda min$, $Zmin \approx \lambda min/2$

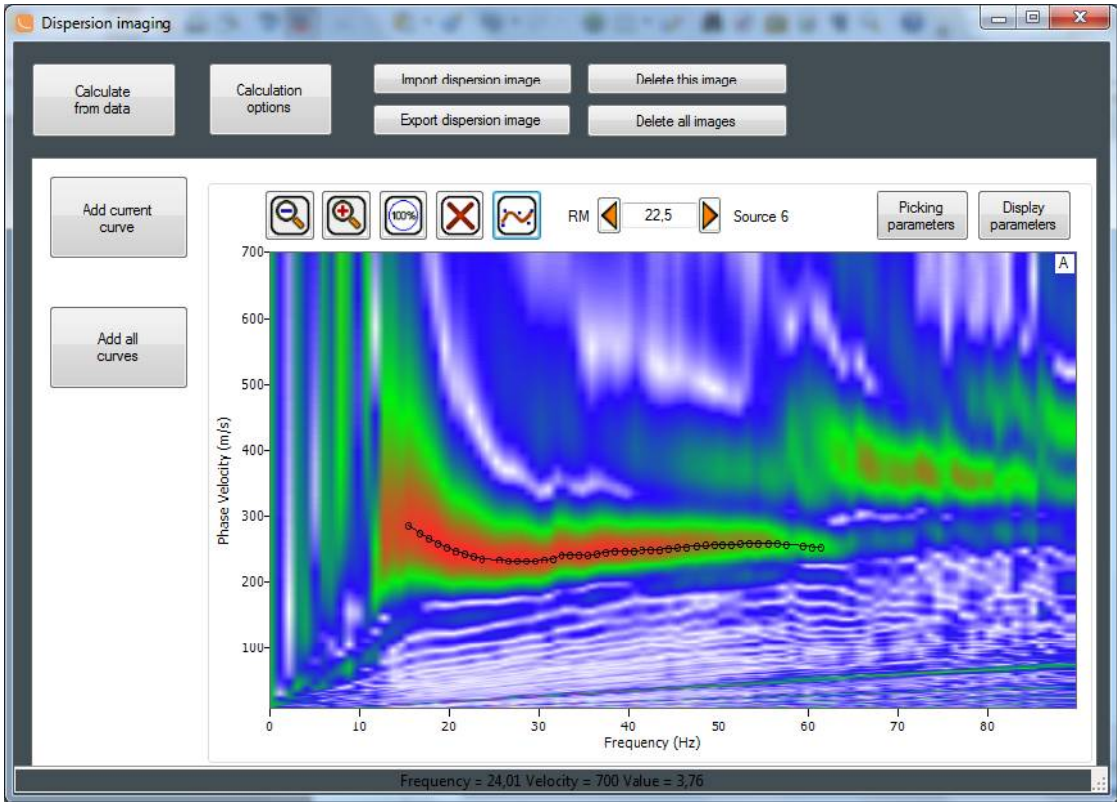
However, in practice the primary factor determining the maximum wavelength is the source. Usually it is the first dozens of meters.

The distance between the source and the first receiver is usually 1-4dx (the method data were taken from *www.masw.com*)

A typical seismogram obtained using the MASW method is shown below:



2) **Dispersion analysis** – building dispersion images. A dispersion image is calculated for each seismogram. A typical dispersion image is presented in the figure below (generated using the MASW module). The calculation procedure is described in the article by Choon B. P. et al. (1998). The dispersion curve is extracted from the image by means of picking by amplitude maximums:



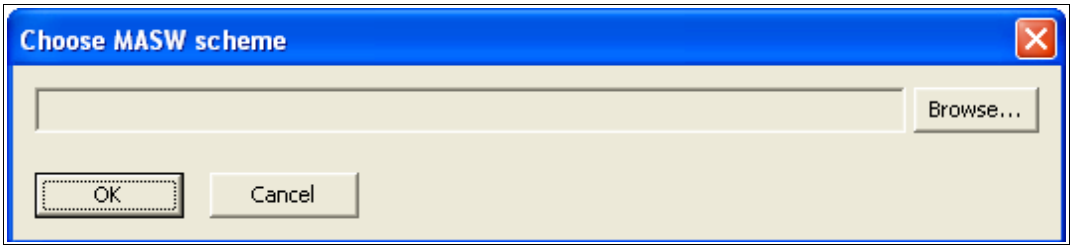
3) **Final step – inversion** – finding the S-velocity profile whose theoretical dispersion curve is as close to the experimental curve as possible. Occam's inversion is implemented in the MASW module – the root-mean-square error between the curves is minimized while maintaining the maximum model smoothness (Constable S.C. et al.(1987)). The S-velocity profile is tied to the midpoint of the receiver spread. A two-dimensional S-wave velocity profile is built by interpolation between the obtained vertical profiles.

Below is a detailed description of the above processing steps for the MASW module in the RadExPro package.

General Procedure of Working with the Module

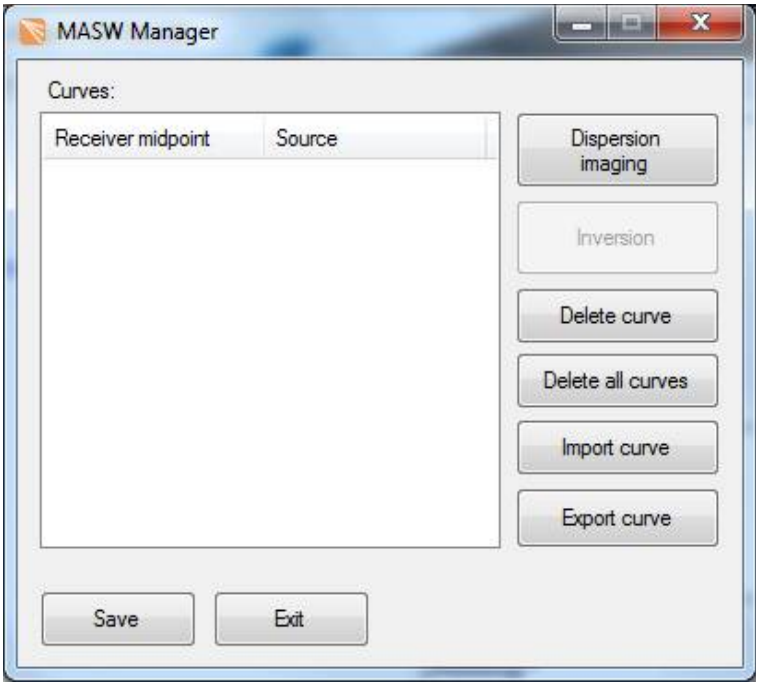
Data are processed within a processing scheme. A scheme is a combination of dispersion images, corresponding curves, the resulting model, and image and model calculation and visualization parameters. Each scheme is stored in a separate MASW directory within the project. This module is a stand-alone module.

When the module is added to the flow, the scheme selection dialog box opens . Create a new scheme or select an existing one by pressing the **Browse…** button:



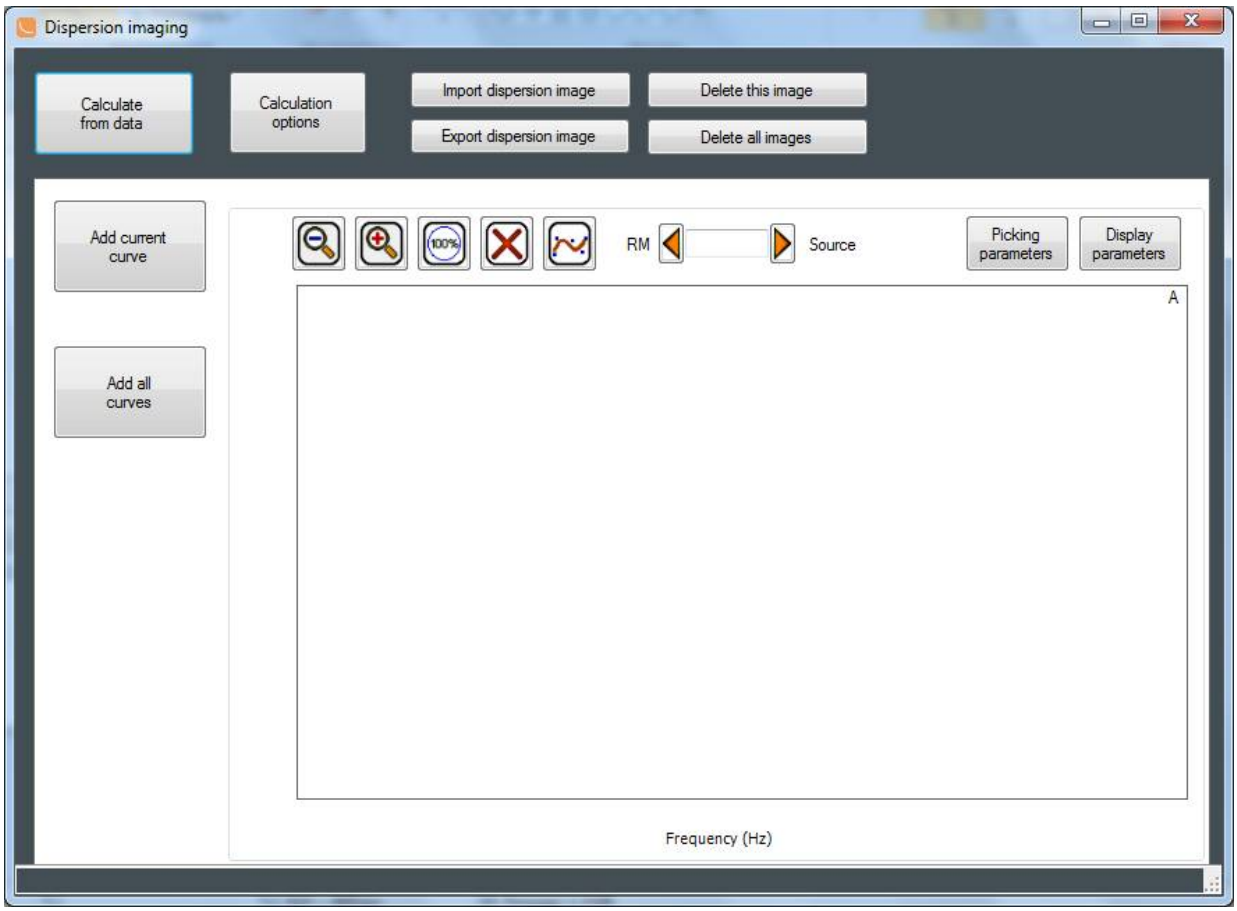
After selecting the scheme, launch the module by pressing the **Run** button. The main project management window – **MASW Manager** – will appear. The left part of this window shows a list of all dispersion curves that have been added to the scheme. The curves (and, therefore, Vs profiles) are tied to the midpoint of the receiver spread and sorted in an ascending order. Source number for these midpoints are shown in the manager as well. This window remains empty until at least one

image is processed and added:



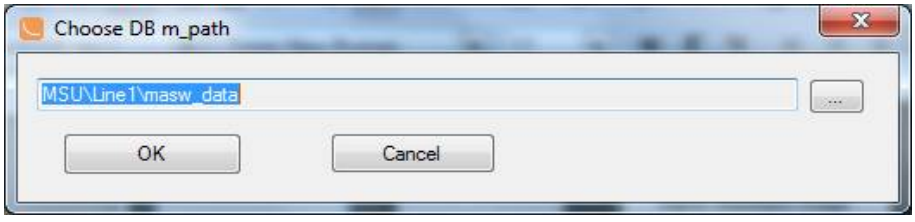
- **Dispersion Imaging**

This menu opens the dispersion image calculation and processing window:



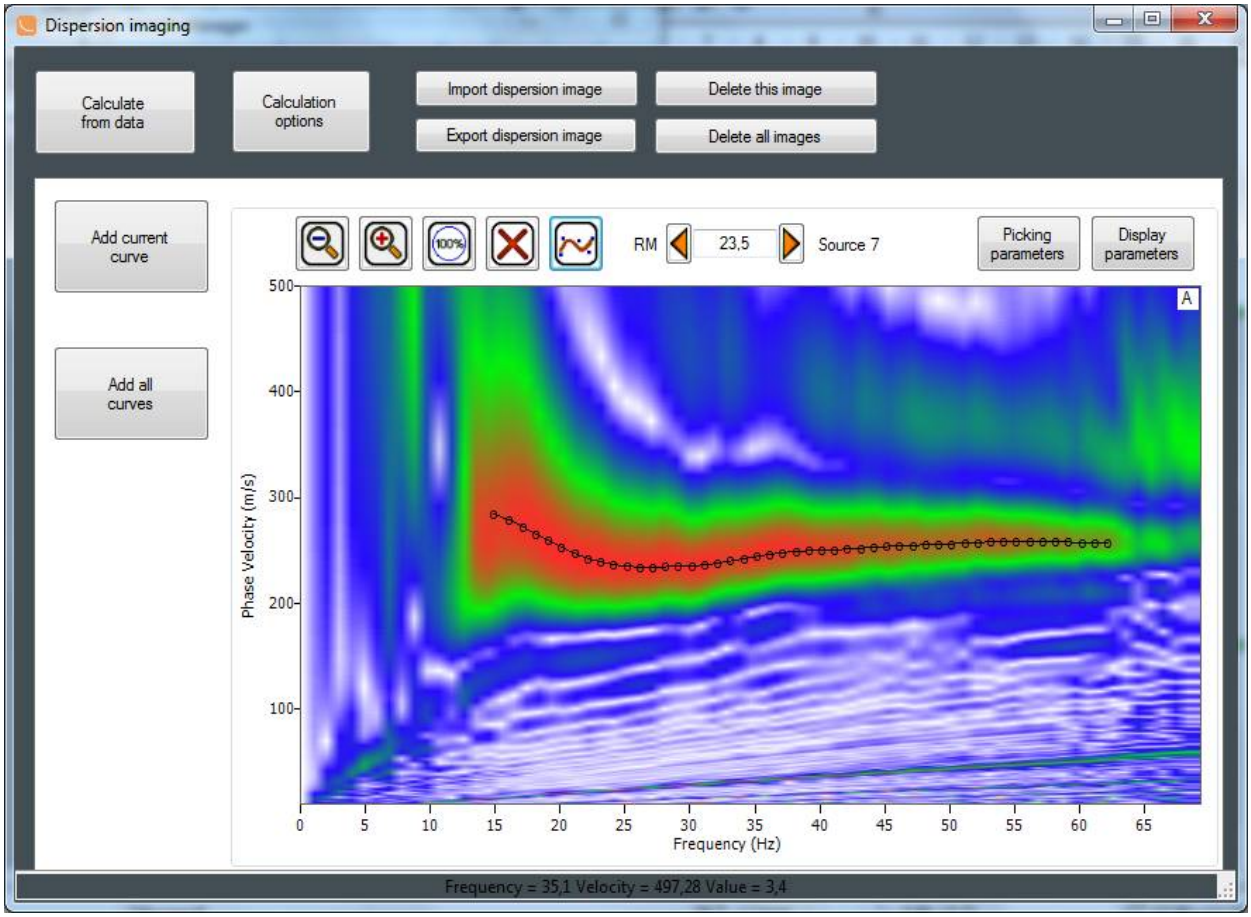
A dataset with assigned geometry is input into this function. Header fields **sou_x, rec_x, offset, ffid,** and **chan** must be filled in for the module to operate properly. Source-receiver geometry is assigned using standard RadExPro tools – Geometry Spreadsheet or Trace Header Math module. The dataset can contain a single seismogram or several seismograms (for example, the entire survey). It is recommended to combine all source points into one dataset to make working with the module easier.

To select the dataset and start the dispersion image calculation, press the **Calculate from data** button, select the dataset and press **OK.**

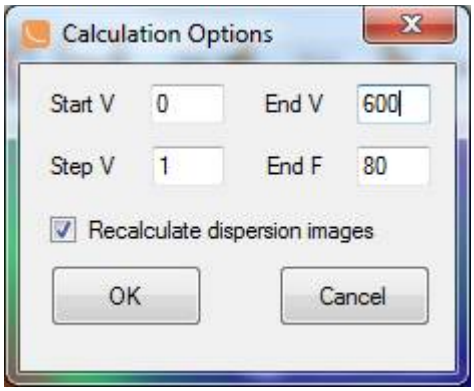


The calculation process will be launched for all source points in the dataset.

As a result, a dispersion image will appear in the **Dispersion Imaging** window:



By default, the image is calculated in the phase velocity range of 0 to 500 ms with 1 ms steps and in the frequency range of 0 to 70 Hz. If the image exceeds the frequency or velocity limits, change the calculation parameters in the **Calculation options** function check *Recalculate dispersion images* function – dispersion images will be recalculated automatically.



The status bar at the bottom of the window shows the current frequency, phase velocity, and amplitude values.

After the dispersion image appears on the screen, the dispersion curve needs to be "extracted" by means of picking the image by amplitude maximums. Automatic picking mode is activated by

default – points are automatically distributed between the first and the last one by amplitude maximums in the specified window with the specified steps. The **Picking parameters** option is used to set the parameters.

After the image is picked, select the next source point by pressing the arrow in the control panel, and repeat the procedure.

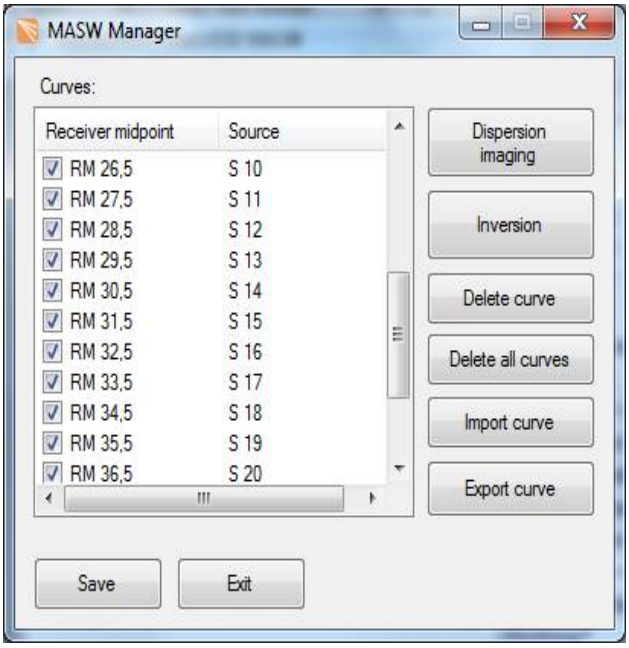
Mouse functions during image picking:

- left mouse button – adds a picking point to the dispersion image;
- right mouse button pressed and held over a point – moves the point around the image;
- double-click with the right mouse button – removes the point;
- CTRL + right mouse button – selects an *area* in which the points will be removed.

When switching from one dispersion image to another, the current picking is saved even if it was not added to the list. This way, the user can always return to any source point, view the picking created earlier, and edit it if necessary.

The calculated dispersion images together with the pickings are parts of the scheme, i.e. if the user exits and then re-launches the project, they will be saved as of the time of exiting.

After all available images are picked, add the curves to the **MASW Manager** list by pressing the **Add all curves** button. Window with added curves is shown below:

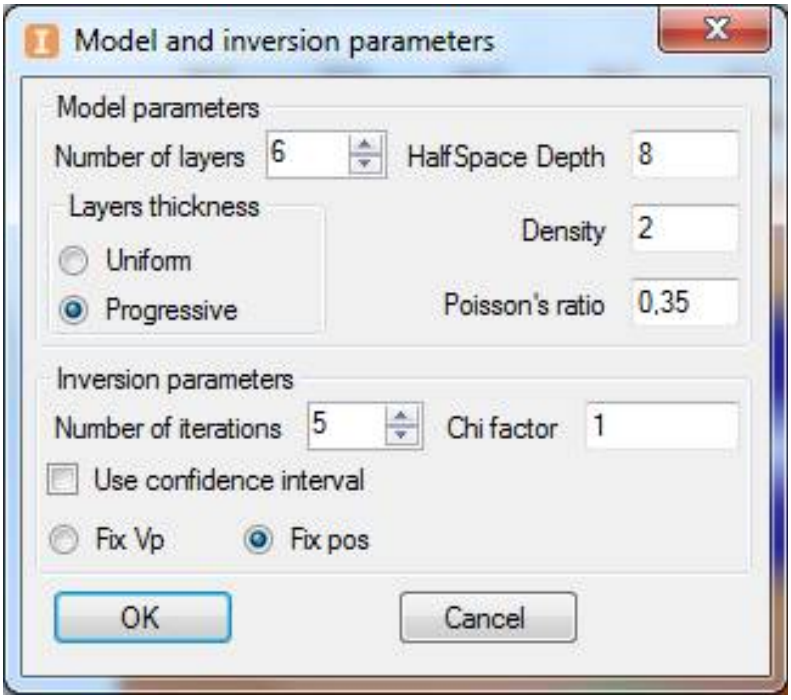


When at least one curve is added to the list, the **Inversion** button becomes active. Now we can move on to the next step – curve fitting and model generation.

Inversion

The initial medium model needs to be specified before the curve fitting process is started. When the Inversion button is pressed for the first time, the dialog box of the initial model filling type selection appears:

It is recommended to select automatic model filling. This will open the parameter setup dialog box:



- **Number of layers** – number of layers in the model;
- **Layer thickness** – type of thickness increase with depth:

  Uniform – uniform layer thickness;

  Progressive – layer thickness increasing with depth;
- **Half Space Depth** – depth to the half space. Approximate estimate of the half space depth should be made based on the maximum wavelength divided by two (Julian I. (2008)).

Automatic filling is performed as follows. The half space depth is broken down exponentially into the specified number of layers depending on the layer filling type. The Vs velocity is assumed equal to Vr/0.88 at each frequency and is averaged between the values occurring within one layer (Jianghai X. (1999)). The Vp velocity is calculated based on Vs and Poisson's ratio using the following formula:

$$\frac{Vp^2}{Vs^2} = \frac{2(1-\nu)}{(1-2\nu)}$$ (Rock physics handbook).

In this case, the same value of Poisson's ratio and density is set for the entire model.

The following inversion parameters are also specified in the current tab:

- **Number of Iterations** – number of curve fitting iterations;

- **Use confidence interval** – this option determines the size of the confidence interval at each frequency (curve picking confidence level). By default, it is disabled, and the value is equal to one;

If this function is enabled, the value will be taken from the one calculated by the dispersion curve (in percent of the picking point amplitude value) (see **Picking parameters** in the **Dispersion Imaging** section).

- **Chi-factor** function – specifies the inversion completion threshold value. When the root-mean-square error becomes less than the chi-factor value during curve fitting, the inversion process is considered to have been completed successfully. By default, this value is equal to one.

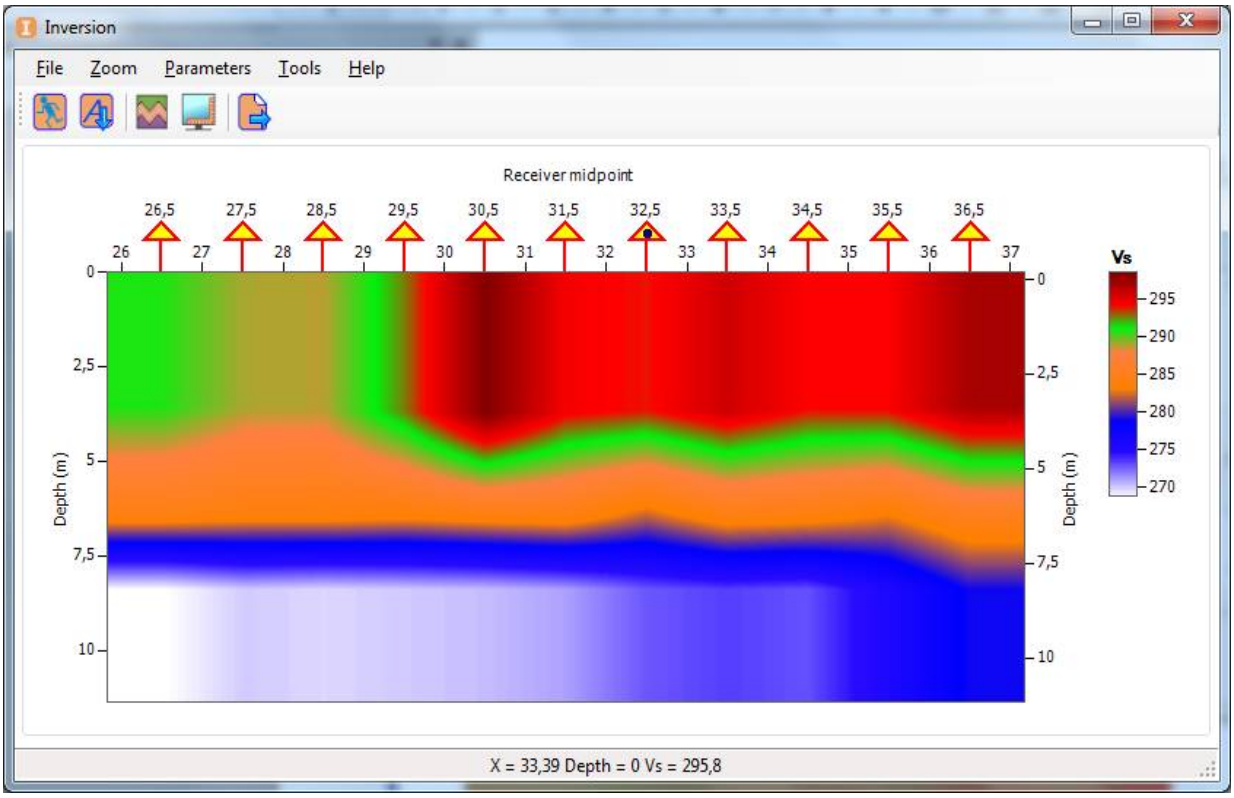
It is recommended to use the default inversion parameters.

When curve fitting is performed simultaneously with Vs fitting, either the Vp value or Poisson's ratio changes. The next option allows specifying which value will be fixed:

- **Fix Vp** – Vp is fixed, Poisson's ratio is variable;

- **Fix Pos** – Poisson's ratio is fixed, Vp is variable;

After specifying all the above parameters, press **OK**. An initial model corresponding to the specified values will appear on the screen:

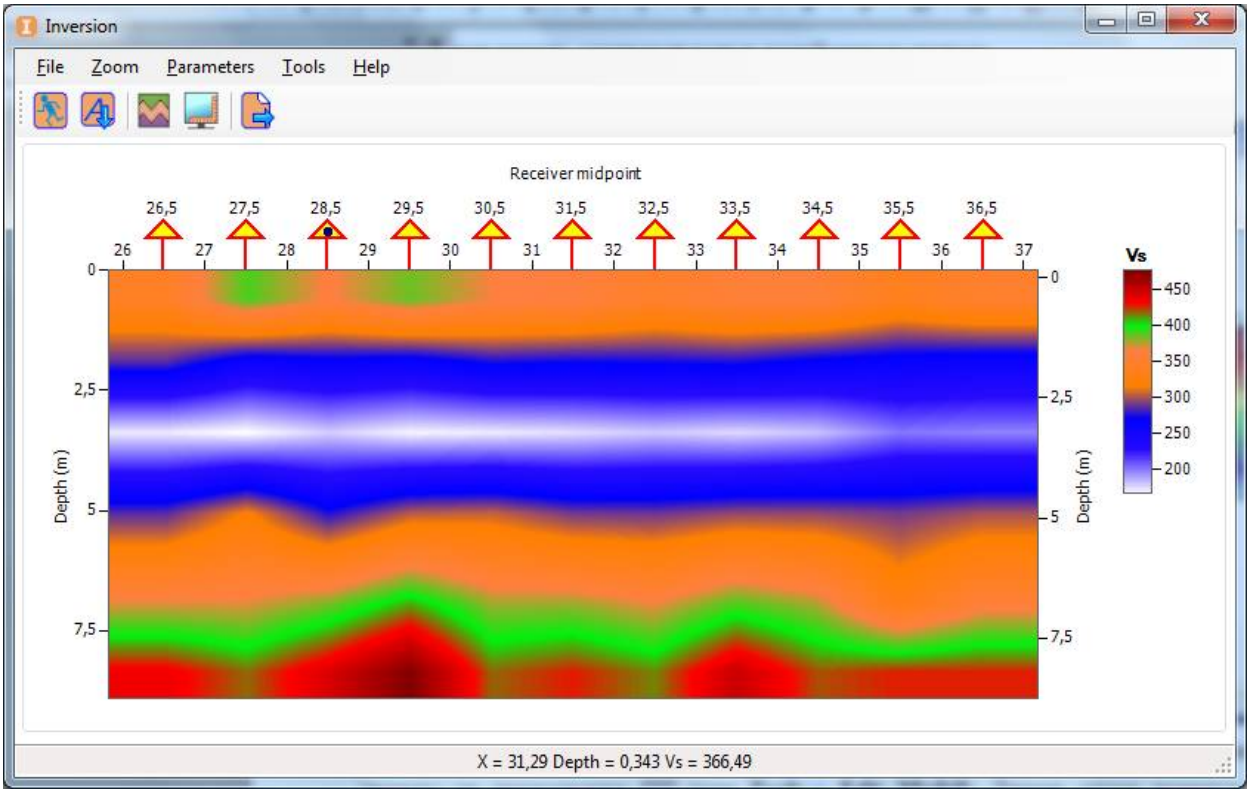


Absolute profile coordinates in meters are shown on the horizontal axis, and depth in meters is shown on the vertical axis.

The receiver spread midpoints to which S-velocity models with the depths are tied are shown as triangles, with coordinates shown above them.

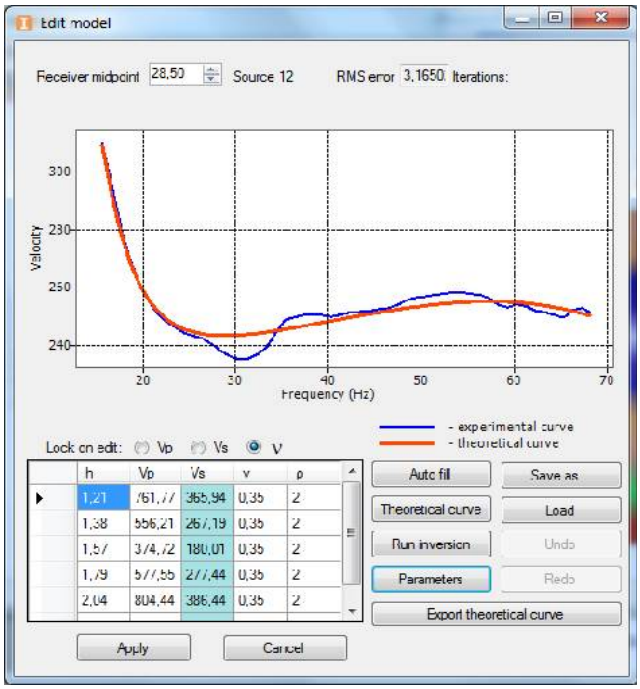
Yellow-filled triangles mean that a certain model is assigned to that point (if automatic initial filling of the layers is used, the model is assigned to all receiver points). White-filled triangles mean than no model is assigned to that receiver point (if the user decides not to use automatic filling – this will be discussed below). This approach to tie-in point designation gives a quick indication of whether any work has been performed with a particular receiver point. The active receiver point whose experimental curve is shown on the **Edit model** tab is marked with a dot in the center of the triangle.

After the initial model is specified, the curve fitting can be launched. The **Tools-> Run inversion** function (also accessible by pressing the button on the tool bar) launches curve inversion for all receiver points. A dialog box indicating that inversion is in progress will appear. As a result of inversion, an updated model corresponding to the fitted curves will be shown on the screen:



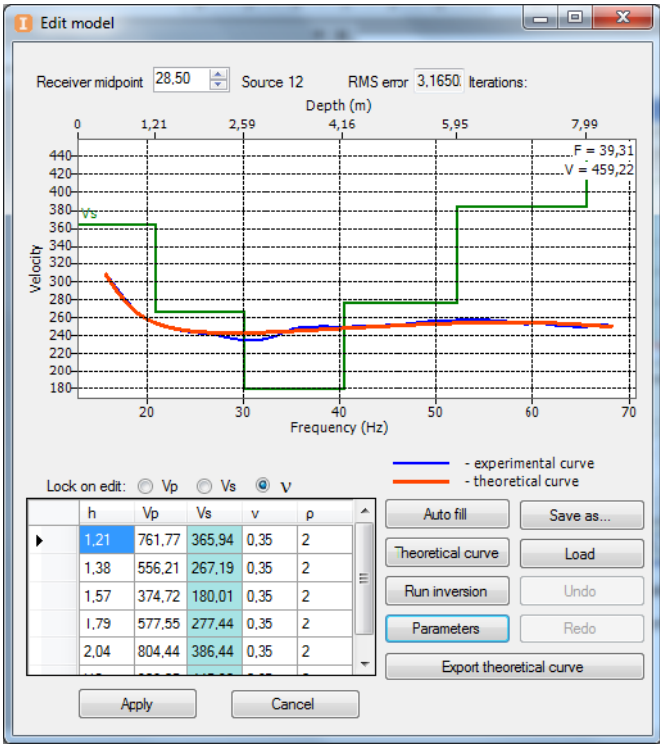
The fitting results for each curve can be evaluated using the **Edit model** option activated by double-clicking the receiver point triangle (or **Tools-> Edit Model**). This option is also a useful tool for detailed analysis of influence of all parameters on the dispersion curve.

Below is shown the model editing window for source point No. 12. It consists of two main parts – image of the experimental and theoretical curve, Vs and Vp models, and a table with model parameters.



The current receiver midpoint number is shown in the **Receiver midpoint** field. The same number is also highlighted as active in the **MASW Manager** window. Root-mean-square error between the experimental and theoretical curve and the number of curve fitting iteration are shown next to the source point number.

The experimental curve is shown in blue, the theoretical one – in red. If necessary, changes of the current Vs and Vp values with depth can be shown in this window (**Parameters** tab):

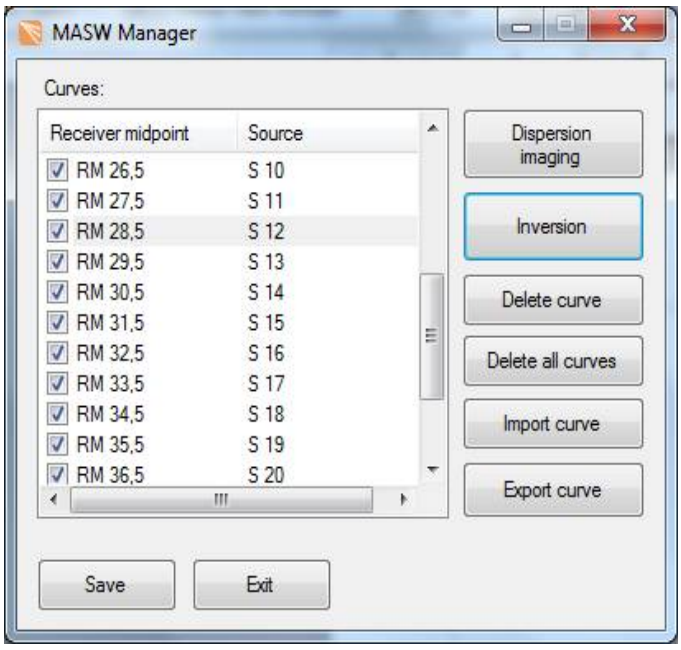


The appearance of the theoretical curve can be changed by changing the model parameters in the table. A detailed description of the available options is provided below.

Detailed Description of Module Functions

MASW Manager

All added curves are shown in the left part of the window. To select the curves that will be used for final model calculation, tick the check boxes next to them. If a model has already been built, the influence of each receiver midpoint on the result can be evaluated by enabling/disabling the source point in this list.



The right part of the window contains the following buttons:

Dispersion Imaging – opens the dispersion image calculation and dispersion curve extraction window;

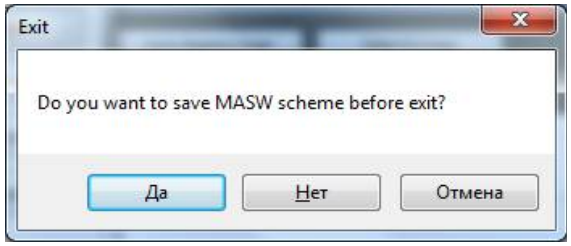
Inversion – generation of the medium model by finding theoretical curves which are as close to the experimental ones as possible. This button remains inactive until at least one curve is added to the list;

Delete curve – removes the curve from the list;

Import curve – loads a curve from a text file;

Export curve – exports the curve.

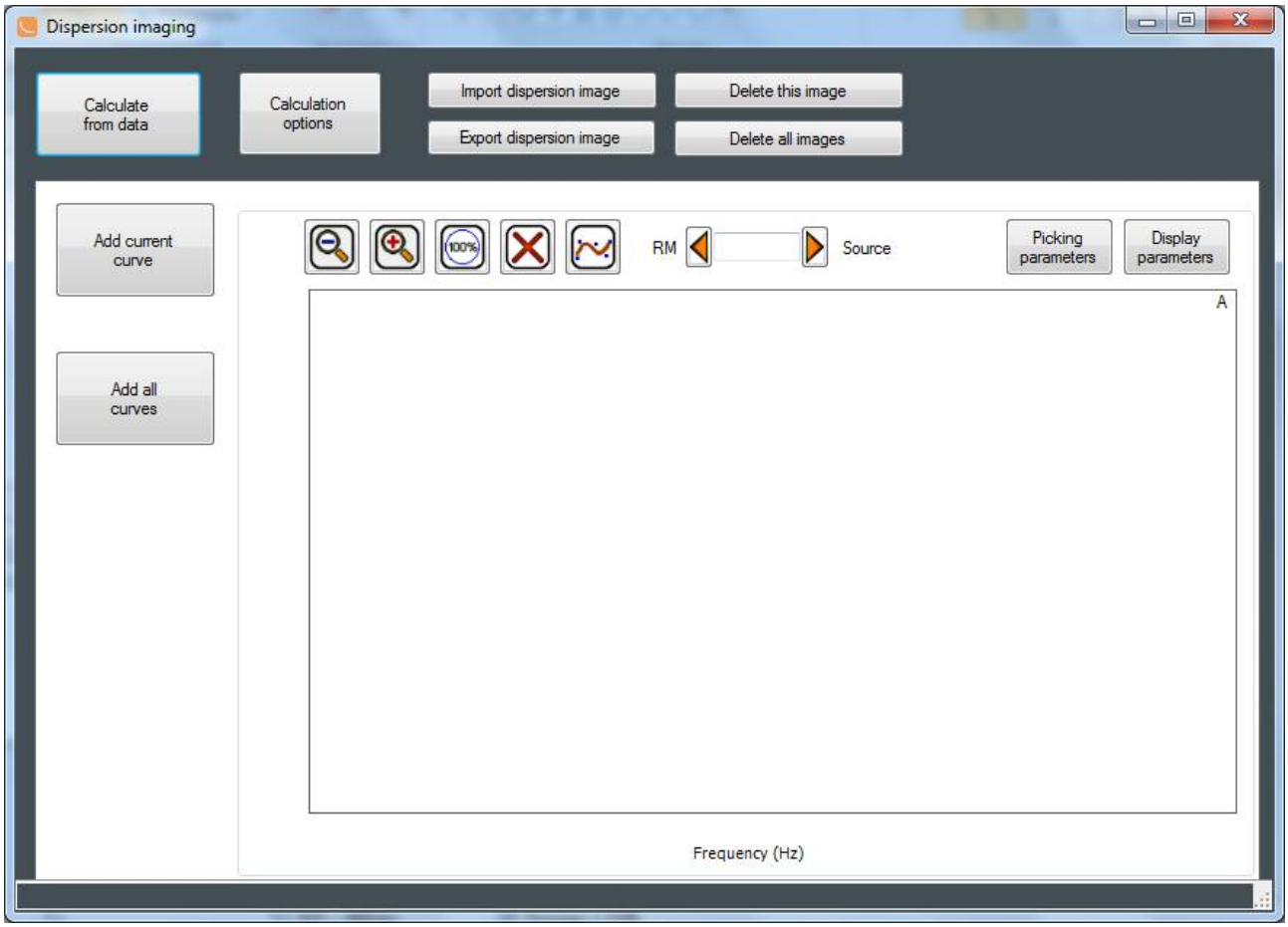
Pressing the **Save** button saves the current scheme. The **Exit** button is used to exit the module; if the scheme has not been saved, the following message will be displayed:



If the user chooses not to save the scheme, all unsaved data will be lost.

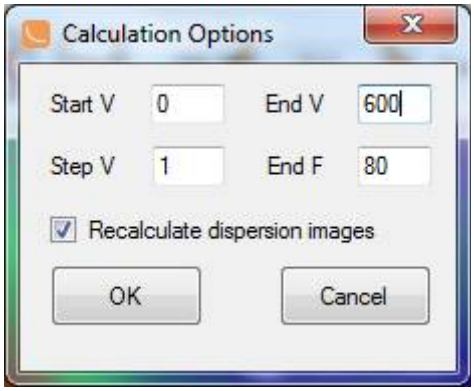
Dispersion Imaging

This menu opens the dispersion image calculation and processing window:



- **Calculate from data** – selects the dataset for dispersion image calculation.

- **Calculation options** – opens the dispersion image parameter calculation dialog box:



Start V – start velocity for the calculation

End V – end velocity for the calculation

Step V – velocity step

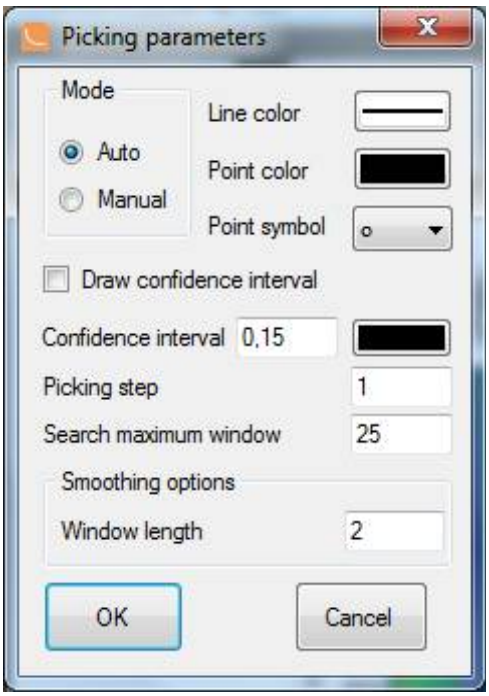
End F – image calculation end frequency

Recalculate dispersion images – dispersion images will be recalculated automatically after choosing OK button

- **Import dispersion image** – imports a dispersion image from a grd file;

- **Export dispersion image** – exports the dispersion image to a grd file.

- **Picking parameters** – this option allows changing the following picking parameters



Mode – automatic or manual picking mode. Automatic picking is enabled by default.

Line color – color of the line connecting the picking points;

Point color – picking point color;

Point symbol – picking point type;

Confidence interval – confidence interval specified in percent of the current amplitude value at the picking point. Used as curve definition confidence parameter during inversion (optional).

Draw confidence interval – enables display of the confidence interval in the dispersion image;

Picking step – automatic picking frequency step;

Search maximum window – window in ms within which the search for the maximum is performed during automatic picking;

Smoothing options – Window length – smoothing window in samples.

- Tool bar functionality:

- zoom out of the image by one level;

- zoom of the image within the selected area;

- revert to original image size;

- delete all points from the image;

- smooth the curve using the base specified in the picking parameters;

- switch between source points.

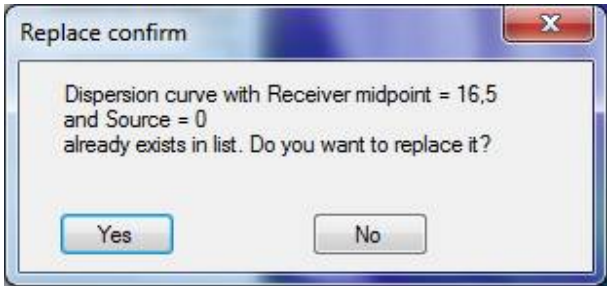
The **Display parameters** option allows editing the dispersion image palette.

The left part of the window contains the following buttons:

Add current curve – adds the current curve to the curve list in the **MASW Manager** window;

Add all curves – adds all existing curves to the list.

If a curve with the same name already exists in the list, the system will display a replacement message:

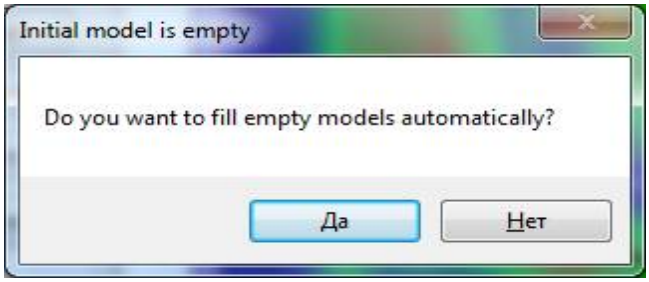

Replace confirm
Dispersion curve with Receiver midpoint = 16,5
and Source = 0
already exists in list. Do you want to replace it?
Yes
No

Inversion

Generation of a model by finding S-velocity vertical profiles at active receiver midpoints.

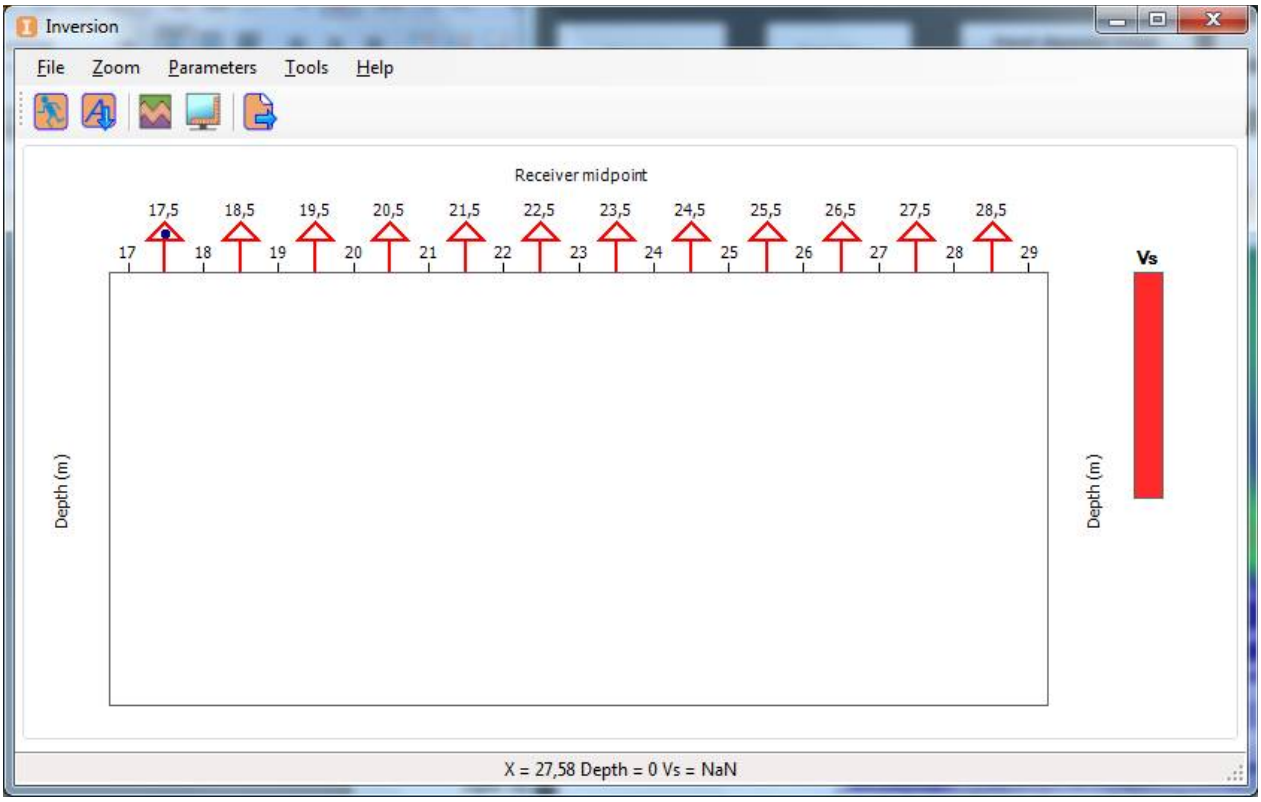
This button becomes active after at least one curve is added to the list.

When the **Inversion** button is pressed, a dialog box opens in the project window prompting the user to select the initial model filling method – automatic or manual:



If automatic model filling is selected, the model parameter setup dialog box will appear. A detailed description is provided in Section 2.2.

If the user decides not to use automatic model filling, an empty model window will appear; all receiver point triangles will be white, meaning that there is no initial model assigned to them:



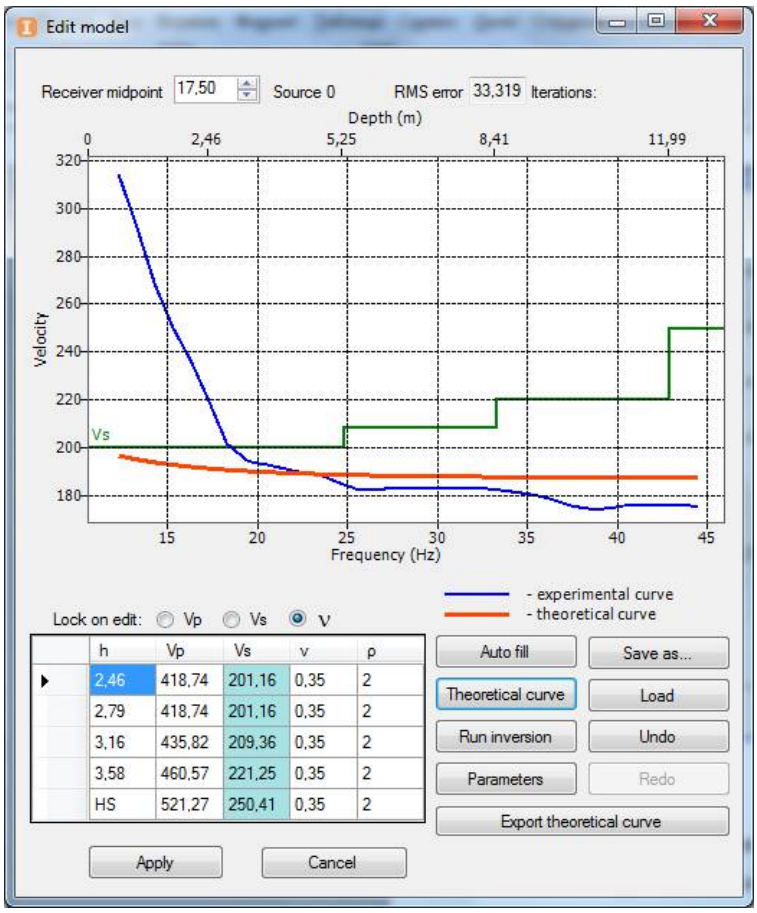
Double-clicking the receiver point triangle opens the curve fitting window (also accessible by selecting Parameters->Edit model). This option allows selecting model parameters for each individual receiver point, and is also a useful tool for detailed analysis of influence of the parameters on the dispersion curve. The initial window of the curve fitting module was shown earlier.

If there is no initial model, it can be filled either manually or automatically by pressing the **Autofill** button. Automatic filling parameters are specified in the main model window: **Parameters-> Model and inversion parameters** (a detailed description was provided above) or

button on the tool bar.

- **Theoretical curve** – shows the theoretical curve corresponding to the model specified in the table.

The theoretical curve corresponding to the model specified in the table is shown in red in the curve display window; the experimental curve is shown in blue. If necessary, changes of the current Vs and Vp values with depth can be shown in this window (**Parameters** tab).When the mouse cursor is hovered over the curve window, the current velocity and frequency values are displayed in the upper right corner.



- **Run inversion** – launches the curve fitting process using Occam's method (the root-mean-square error is minimized while maintaining the maximum model smoothness). Curve fitting is performed by changing the Vs velocity (the table column with the parameter being selected is highlighted in light blue). During this process, either Vp or Poisson's ratio is also updated depending on the inversion parameters.

During each iteration, the theoretical curve and the velocity change with depth (if enabled) are updated in accordance with the current model. The inversion process stops when the number of iterations specified in the parameters is reached or when the root-mean-square error reaches a value below the one specified in the inversion parameters.

After the curve fitting process is completed, the user can perform a more detailed analysis of the theoretical curve by changing other parameters in the table. To display the updated theoretical curve after changing any of the parameters, press the **Theoretical curve** button.

- **Working with the table**

The table contains four parameters that allow changing the shape of the theoretical curve – Vs, Vp, ρ, ν.

The **Lock on edit** switch performs the following function: the value of the selected parameter is locked, and the table column becomes unavailable for editing. If one of the unlocked parameters is changed, the other one will be updated in accordance with the current value of the locked parameter.

- **Parameters –** image parameter setup

Show grid – show lines perpendicular to the axes with the specified step

Tick step for X axis – value step on the X axis

Tick step for Y axis – value step on the Y axis

Show Vp – show the model of Vp change with depth (brown color)

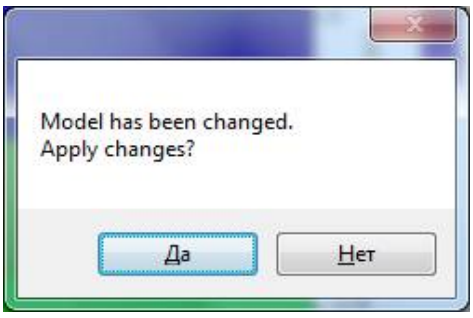
Show Vs – show the model of Vs change with depth (green color)

- **Undo** – undo one step in the value table;
- **Redo** – redo one step in the value table;
- **Save as..** – save the current value table to a text file;
- **Load** – load a value table from a file**.**

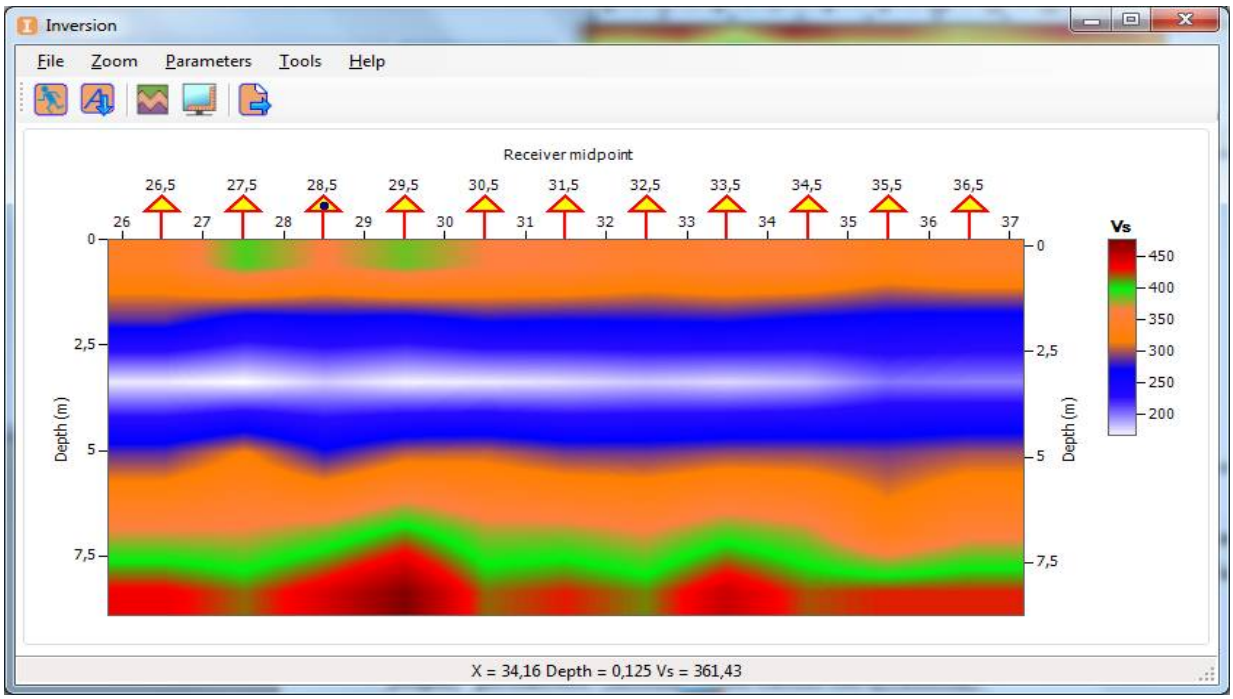
When the **Apply** button is pressed, the model corresponding to the receiver point is applied in the main model window.

The **Cancel** button cancels all changes and closes the model editing application.

Switching between source points is performed either by pressing the arrows or by selecting the receiver point in the model window or the **MASW Manager** window. If the model has been changed but has not been applied, a message prompting to apply the model is displayed:



- **Model Display Window**



- **File**

Export (button on the tool bar) – exports the model to a grd file;

Exit – closes the model window;

- **Parameters**

Edit model – opens the curve fitting window. A detailed description was provided above;

Model and Inversion parameters (button on the toolbar) – opens the model and inversion parameter window (a detailed description of the parameters was provided above);

Display parameters (button on the toolbar)

Palette – model palette selection

Axes – axes properties (steps on the axes)

Display mode – switching between layer-by-layer and grid type of model display.

- **Tools**

Run inversion *(button on the toolbar)* – launches the inversion process for all active receiver points;

Autofill all *(button on the toolbar)* – automatically fills the model values for all receiver points using the parameters specified on the **Model and Inversion parameters** tab. Previously filled values for all receiver points are automatically rewritten!

- **What to remember when building models.**

Vertical velocity profiles are tied to the receiver spread midpoint. Values between the profiles are interpolated.

If a new receiver midpoint is added to an already built model, interpolated values at the point where it was added are used as the initial values for the model of that source point. The receiver point remains white, since no model editing took place at that point.

When a receiver point is right-clicked, the **Spread this model for all shotpoints** option is displayed. This option applies the current model to all receiver points.

Influence of each receiver point on the model can be evaluated by enabling/disabling receiver points in the **MASW Manager** window. When a receiver point is unchecked, it will be removed from the model and will not affect the end result.

Bibliography:

1) Choon B. P., Richard D. M., and Jianghai X. (1998) "*Imaging dispersion curves of surface waves on multi-channel record"* SEG Expanded Abstracts

2) Constable, S.C., Parker, R.L., and Constable, G.G. (1987). "*Occam's Inversion: A Practical Algorithm For Generating Smooth Models From Electromagnetic Sounding Data"* Geophysics, 52, 289-300.

3) Gary M., Tapan M., Jack D. "*The Rock Physics Handbook"*

4)Julian I., Richard D. M., Kansas Geological Survey, George T., CReSIS, The University of Kansas (2008) "*Some practical aspects of MASW analysis and processing*" SAAGEEP Extended

5)Jianghai X., Richard D.M., and Choon B.P., (1999) “Estimation of near-surface shear-wave velocity by inversion of Rayleigh waves” Geophysics, 64, 691-700

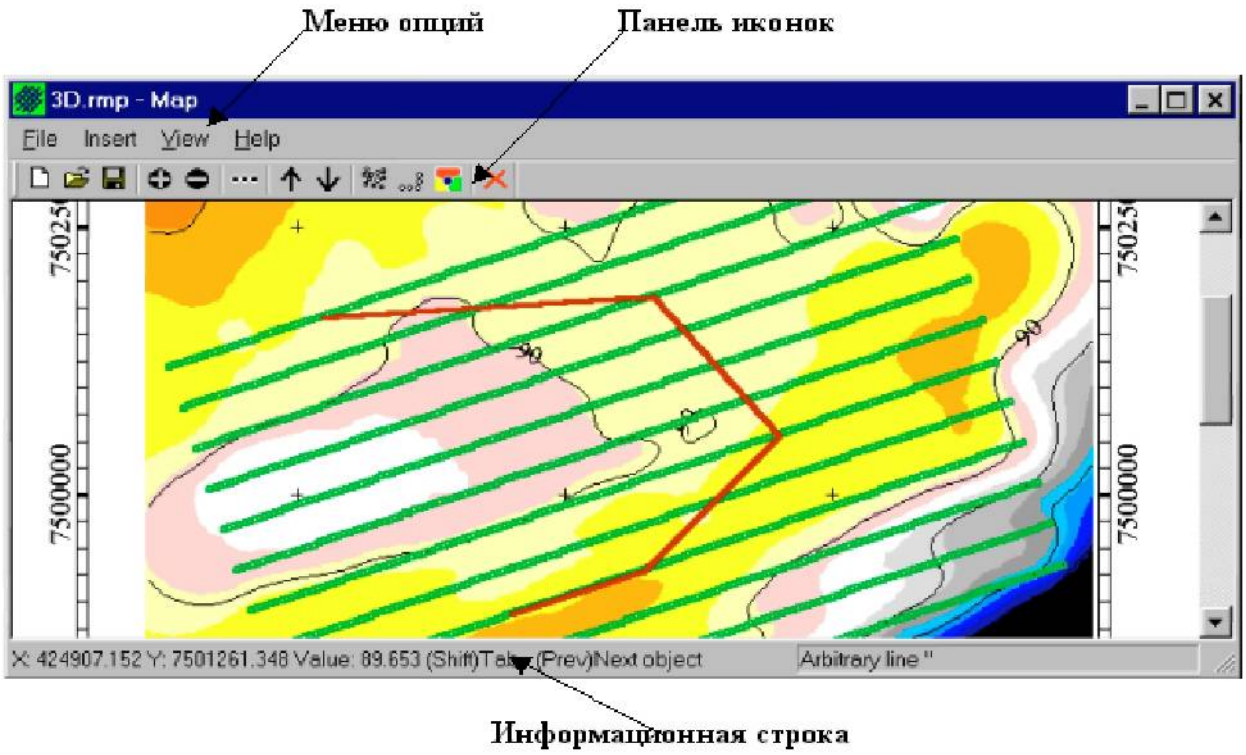
6)Rix G.J. (1988) “Experimental study of factors affecting the Spectral-Analysis-of-Surface-Waves method”, PhD Diss., Un. of Texas at Austin

7)www.masw.com website by Park Seismic LLC

Special tools

Map

To activate the **Map** application, select the **Tools/View Map** command in the primary **RadExPro** window (project tree window) or double-click the left mouse button (**MB1**) on one of the tree sections of **Area** level.



The menu and tool bar

- **File** contains the following commands:

- **New** command creates a new **Map.** When selecting this command the user will be offered to save the current **Map** image,

- **Open...** command opens a standard file opening window. Here, select the previously saved map file *rmp.

- **Save** command saves the current **Map** image on disk in *rmp file,

Save as... command opens a standard file saving window to save the current **Map** image on disk in *rmp file under specified name.

Print... command sends the Map image to printer,

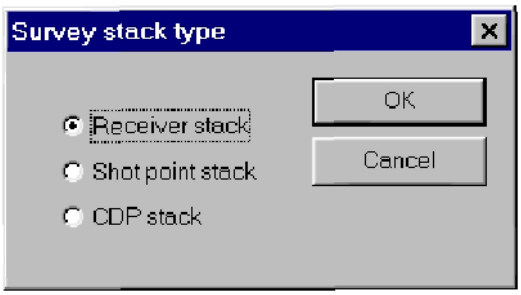
Print Preview command enables image preview before printing,

Print Setup... command opens a standard window for printer and printing parameters settings,

Exit command allows exiting the application. If the current map was changed then when selecting this command the warning about the necessity of saving the changes will appear.

Insert contains the following commands:

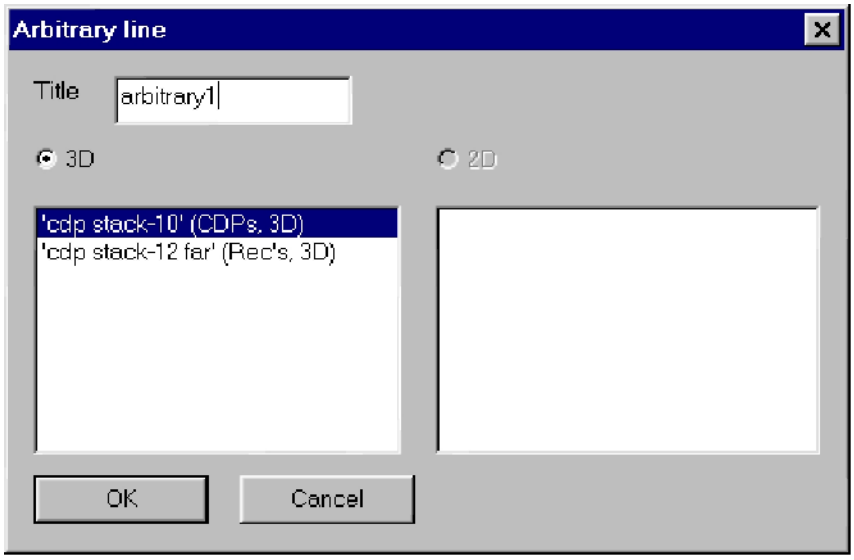
Survey...— command opens the **Choose dataset** window where you can select a stacked dataset registered in the project which should be added to the current **Map** image as a new line-object. As soon as the dataset is selected the **Survey Stack Type** dialog box opens:



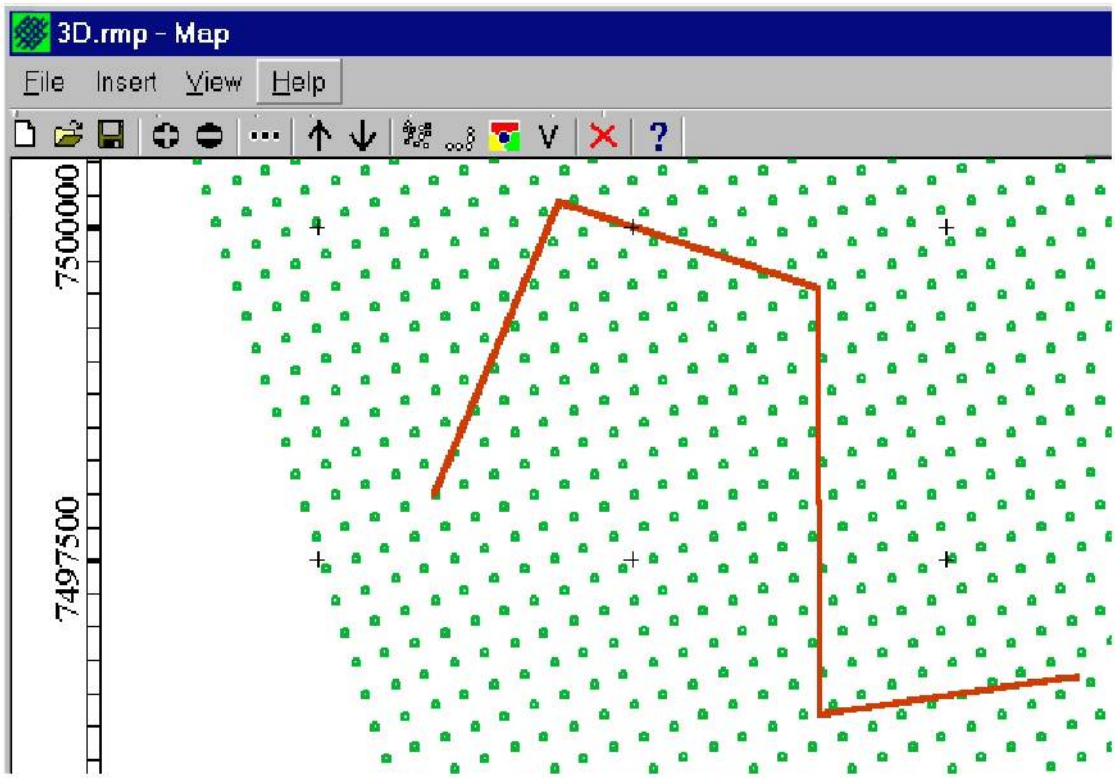
When the stack type of the line is selected,, click the **OK** button. Depending on the stack type choice, for displaying the new line on the map the application will use values of either CDP_X/CDP_Y or SOURCE_X/SOURCE_Y or REC_X/REC_Y fields. Besides, for *2D* data, fill in the following fields: ILINE_NO (=[CDP] or =[SOURCE] or =[FFID] depending on the stack type choice) and XLINE_NO=1.

Arbitrary Line... command opens the dialog box for arbitrary line selection. This opportunity can be used in order to enable combining of fragments of several intersecting lines in one and the same **Screen Display** window. In the Title field the title for arbitrary line should be specified. In the 3D or 2D field from the list of available objects, select those lines, which will be displayed on

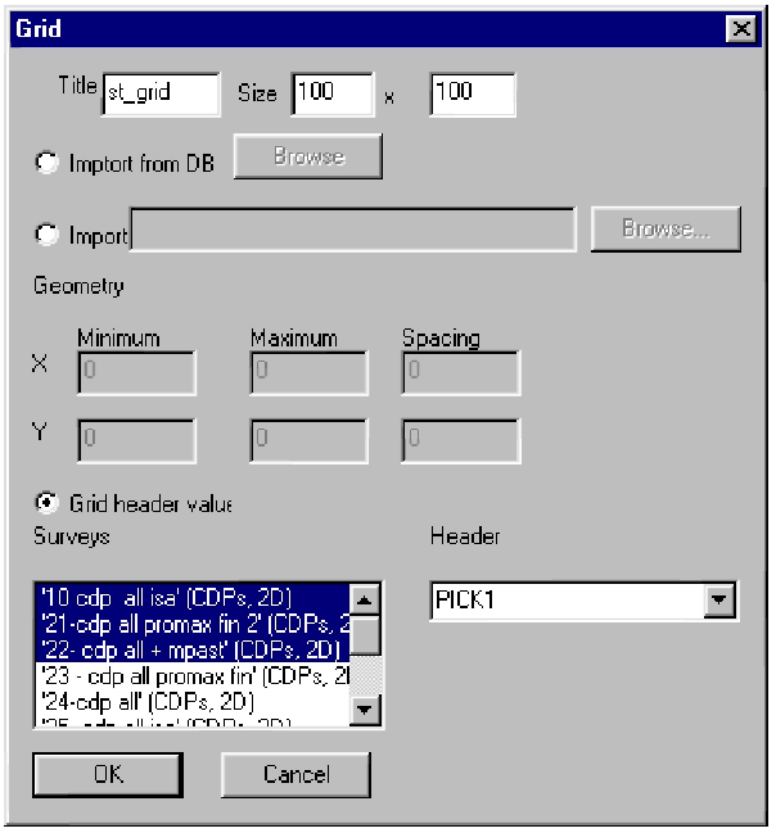
screen and to which the arbitrary specified line will be related.



When all parameters are specified,, click the **OK** button. After that you can assign an arbitrary line on the map via the left mouse button (**MB1**). You can exit the mode of arbitrary line specification by clicking the right mouse button (**MB2**). When double-clicking the left mouse button on the arbitrary line it will be visualized in the *Screen Display* module window.



Grid... opens the dialog box for grid parameters,



that contains the following items:

- **Title** sets the grid title,

- **Size** sets the grid size in cells,

- **Import from DB** imports the file from the database,

- **Import** imports the file of ASII format from the disk containing XYZ columns. The **Browse...** button opens a standard box for file selection,

- **Geometry** This item is available if the **Import** item is selected. When the file which to be imported is selected the maximum and minimum values of X and Y and the grid cells dimensions in X and Y directions should be specified in the **Geometry** item,

- **Grid header value** allows construction of a grid on the basis of dataset coordinates and selected header field value,

- **Surveys** shows the datasets added to current **Map** image. You can include/exclude it from the list

by clicking the left mouse button (MB 1).

- **Header** shows the list of current project headers. Click the left mouse button (**MB1**) to select the header from which the value for the grid will be taken.

View

- **Tool Bar** shows/hides the tool bar,

- **Status Bar** shows/hides the status bar

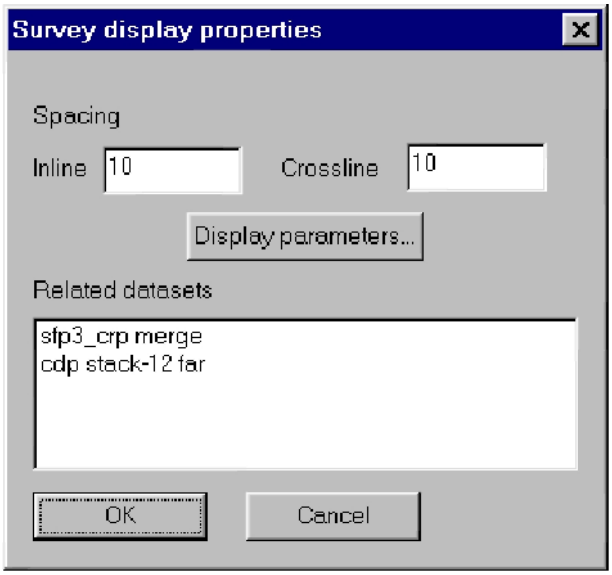
On the top of the client area of the window there is a tool bar containing several commands that are not accessible through the menu:

- **Zoom in** enlarges the map image

- **Zoom out** diminishes the map image

- **Properties** opens the properties dialog of the the currently active object on the **Map**

When a survey object is active the **Survey display properties** dialog box opens:



It contains the following items:

- **Spacing** allows entering the inline and crossline step marks on map. The marks are displayed as circles,

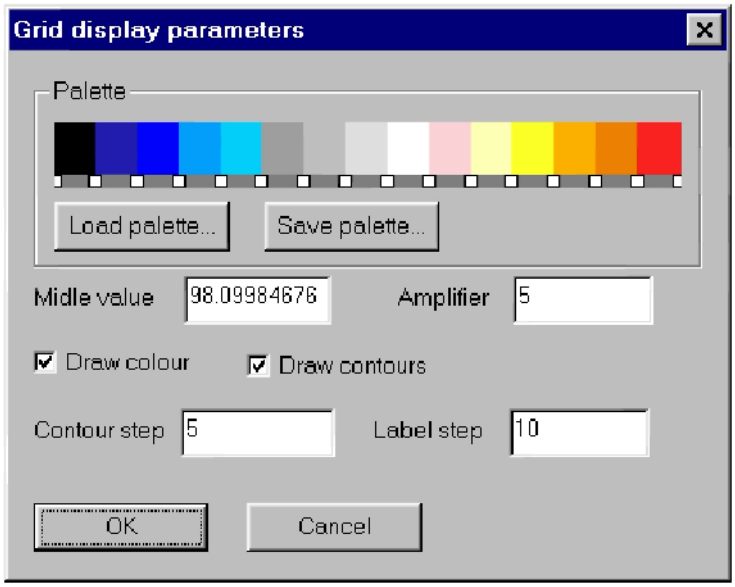
- **Display parameters...** opens the **Display parameters** dialog box for the **Screen Display** module parameters selection,

- **Related datasets** enables selection of dataset from the database which will be displayed in the **Screen display** module window together with active **Map** object.

- The **Ins** key activates the dialog box for database object selection.

When an arbitrary profile object is active, the **Screen Display** module dialog box appears.

When the grid object is active the dialog box for grid displaying appears:



It contains the following items:

- **Palette** In this field the current palette used for grid displaying is shown. The pallet should be set in the form of set of points with specified colors. Colors are linearly interpolated between the set points. You can create, replace or delete points, change points' color. The points are situated under palette image on the gray bar (white rectangles).

- **To change a point position,** click on it by the left mouse button (**MB1**) and, holding it, drag to the required position and release the button. The pallet image will be changing while replacing.

- **To change the color of an active point,** double-click on it by the left mouse button (**MB1**). A standard dialog box for points color changing will open.

- **To create a new point with specified color,** click on the required place by the left mouse button simultaneously pressing on the Shift key (Shift+**MB1**). A standard dialog box for point color selection will open.

- **To delete a point from the palette,** click the right mouse button (**MB2**) on it.

- **Load palette...** opens a standard window for file opening to enable selection of RGB ASII file form disk,

- **Save palette** ... opens a standard window for file saving to save the current palette as RGB ASCII file on disk,

- **Middle value** shows the grid values corresponding to zero level of the color range (it is shown in **Palette** item). Usually it is mean value of palette values. Values can be edited by displacing the zero color level,

- **Amplifier** multiplies the grid value on the value specified in this field,

- **Draw color** allows displaying or not displaying the values on grid in color on **Map** image,

- **Draw contours** allows displaying or not displaying grid contours on **Map** image,

- **Contour step** allows contour step displaying on **Map** image,

- **Label step** allows displaying the step of contour text marks displaying on **Map** image,

- **OK** allows exiting the window with saving the changes,

- **Cancel** allows exiting the window without saving the changes,
- **To front** displaces the active map object to front,
- **To back** displaces the active map object to back,
- opens the **Interactive Velocity Analisys** application,
- **Delete object** deletes active object of the map,
- **Help** opens the **Help** application.

Status Bar

At the bottom of the running window there is a status bar:

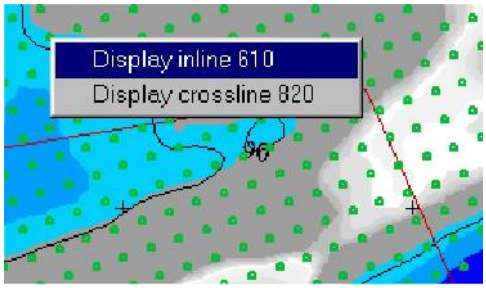
In the left part of the status bar there are coordinates of cursor position point and the hint with available mouse button commands. In the right part of the status bar there is a name of the active object.

Keyboard and commands

Tab/Shift+Tab make the next/previous map object active.

If an active object is an arbitrary profile then by double-clicking the left mouse button (**MB1**) you can open it in the **Screen Display** module.

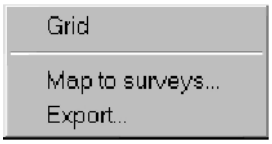
By clicking the right mouse button **(MB2)** the window for X- or Y-profile selection can be opened.



After X- or Y-profile has been activated the **Display Parameters** window opens. After all required

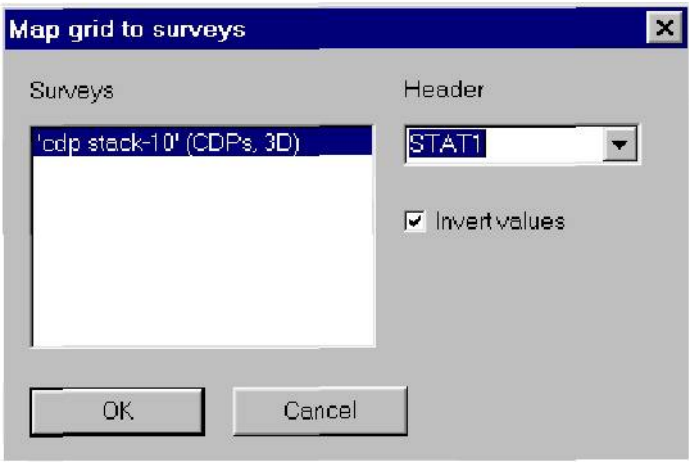
parameters have been specified the profile opens in the *Screen Display* application.

If an active object is a grid then by clicking the right mouse button **(MB2)** you can activate the pop-up menu of the grid,



which contains the following options:

- **Map** to surveys... opens the **Map grid to surveys** (grid saving into seismic data header fields) dialog box. From the **Surveys** and **Header** list

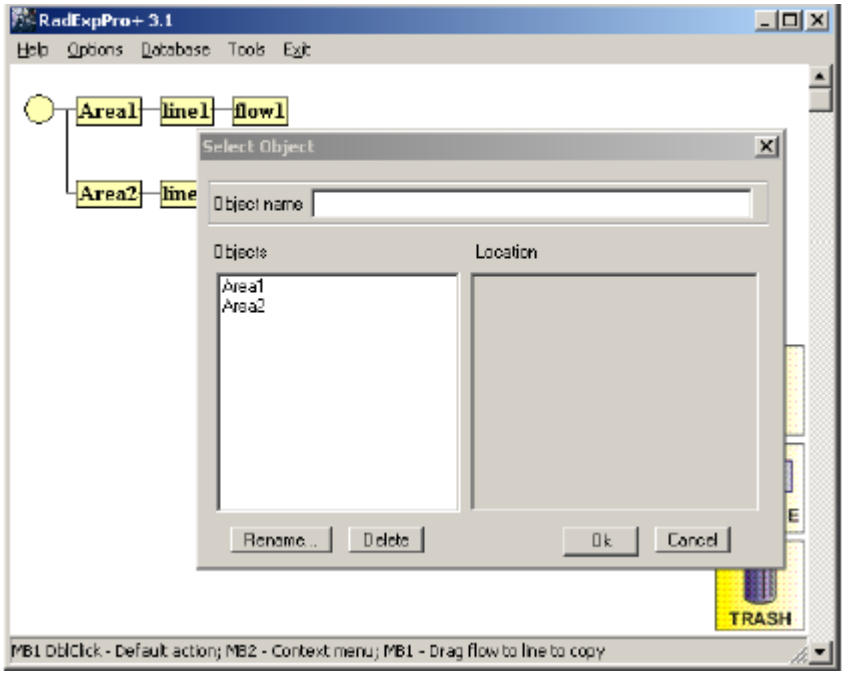


you should select available dataset and respective header to which the grid values will be saved. If you desire to change the sign of grid values to be saved, select the **Invert values** item.

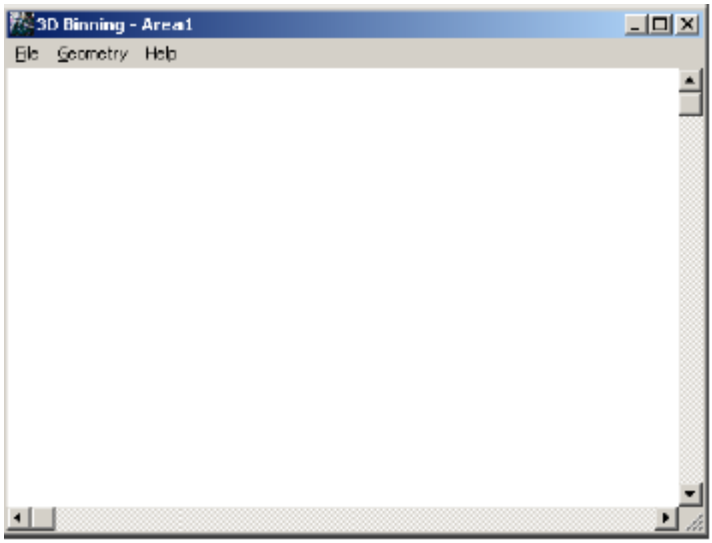
- **Export** opens a standard dialog box for file saving to save the grid values on disk in the file of ASII format in form of XYZ columns.

3D CDP Binning

The application is designed for 3D CMP data binning. To run the **3D CDP Binning** tool choose the corresponding command from the **Tools** menu. A dialog window for area selection, to which the binning refers, opens (element of a database of the *Area* level).



The principal tool window **3D CDP binning** opens after choosing the area**.**



The work with the tool is carried out via the principal menu, containing 3 items:

File, **Geometry** and **Help**.

Menu **File** includes the unique command **Exit** for exit from the application.

Menu **Geometry** includes the following commands:

- **Initialize geometry** – initializes the internal geometry table, attached to the selected area from the database. This table includes the coordinates of CMPs and parameters of binning mesh.

- **Assign geometry from dataset** – this command opens the dialog window for dataset selecting, which geometry will be used for creating the tables of middle points. The following header fields should be filled in correctly in the dataset: TRACENO, REC_X, REC_Y, SOU_X, SOU_Y. The middle point coordinates (common middle point tables) are calculated for each trace when creating geometry table:

X = (REC_X + SOU_X)/2

Y = (REC_Y + SOU_Y)/2

These coordinates are kept in the middle point table and are not saved to the dataset headers.

- **CDP binning** – when you choose the command, the application goes to the binning mode. The main window represents common middle points for the dataset, which were selected from the previous command, and binning mesh. Moreover, a dialog window of the mesh parameters' options opens along with tool bar that allows selecting the scale of the image and editing the binning mesh manually via the mouse. The following paragraph contains more details on binning mode.

- **Assign CDP's to dataset** – the last command of the menu **Geometry** that allows assigning the binning results of this or that dataset. A dialog window for selecting the dataset opens when you choose the command, the binning results will be written to the trace headers of this dataset. The following fields will be filled in:

ILINE_NO – number of binning cell in horizontal scale, into which the trace has fallen

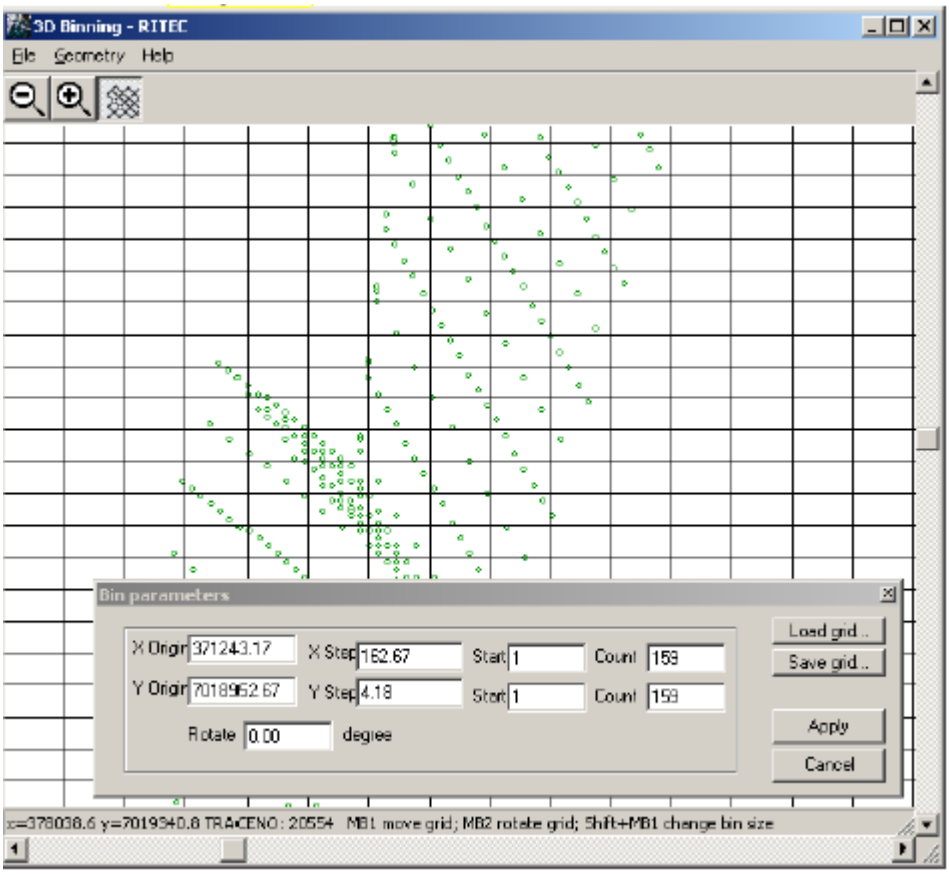
XLINE_NO – number of binning cell in horizontal scale, into which the trace has fallen

CDP = ILINE_NO + XLINE_NO * 10 000

CDP_X и CDP_Y – coordinates of the binning cell center (bin), into which the trace has fallen

Working in binning mode

To go to the binning mode select the command of the menu **Geometry/CDP binning**. The common middle points of the selected dataset (as green samples) and binning mesh will be reflected in the main application window.



Now the dialog window for selecting the mesh options opens along with the tool bar that allows changing the image scale and interactive editing of the binning mesh using a mouse. The dialog window **Bin parameters** allows adjusting binning mesh parameters. The window contains the following fields:

- **X Origin** . X-coordinate of the origin (left lower angle) of the mesh.

- **Y Origin** . Y-coordinate of the origin (left lower angle) of the mesh.

- **X Step** . horizontal size of a cell (bin).
- **Y Step** . vertical size of a cell (bin).
- **Count** . number of cells in horizontal /vertical scale correspondingly.
- **Rotate** . clockwise rotate mesh angle in degrees.

Buttons **Save grid** and **Load grid** allows saving the mesh parameters as a database object and load the bin mesh saved earlier.

Buttons **Apply** and **Cancel** close the work in the binning mode. If the button **Apply** is pressed, the binning parameters are saved to the internal geometry table. If the button **Cancel** is pressed, the last changes are canceled and the table is not overwritten.

- ## Toolbar

The tool bar in the upper part of the screen contains three buttons:

- increasing the size of an image fragment. To increase the size of an image fragment click this button, click the left mouse button on one of the angles of the selected fragment, move the cursor to the opposite angle of the fragment, and click the left mouse button. The selected fragment will be increased to fit the window size.

- decreasing the image scale to initial size.

- Toggling on/off interactive editing binning mesh parameters mode using the mouse. You can perform the following operations in this mode:

- **Moving the binning mesh** - is performed with the left mouse button
- **Rotating the binning mesh** - is performed with the right mouse button

- **Squeezing** / stretching the binning mesh due to bin size changing - is performed with the left mouse button holding the Shift button. All interactive binning mesh parameters changes, are reflected in the fields of dialog window options.

Sequence of work with the application

- If 3D binning is performed for the 1st time for the given area (database object of the *Area* level), you have to create (initialize) the geometry table, linked with the selected area, after having run the application **3D CDP binning**. To do so use the menu command **Geometry/Initialize geometry**.

- Select the dataset, the CMPs of which are going to be reflected in the application main window in the binning mode. To do so use the menu command **Geometry/Assign geometry from dataset**.

- Use the menu command **Geometry/CDP binning** to go to the binning mode. You will have a binning mesh linked with the area, reflected in the window, as well as CMPs of the dataset selected from **Geometry/Assign geometry from dataset**. When you finish the editing of binning parameters click the **Apply** button in the dialog **Bin parameters**. The binning mesh parameters will be saved to a table of the selected area.

- In order to apply the binning mesh to any dataset, choose the menu command **Geometry/Assign CDP's to dataset**. When you call this command several times you can perform binning of several datasets on the selected mesh.

- You don't need to carry out the steps 1 and 2 when you call the application for the next time for the same area. A previous binning mesh along with the CMPs of the last selected dataset will be reflected when you use the menu command **Geometry/CDP binning** in the application window. You are free to edit the bin parameters and after their application, perform the binning of different datasets (see paragraphs 3 and 4). To reflect the CMPs of another dataset with the same binning mesh, use the menu command **Geometry/Assign geometry from dataset**. To remove the existing binning mesh and create a new geometry table, choose the menu command **Geometry/Initialize geometry**. After that you need to carry out all the steps from 2 to 4 in series.

Tape Loader (program of reading data from tapes)

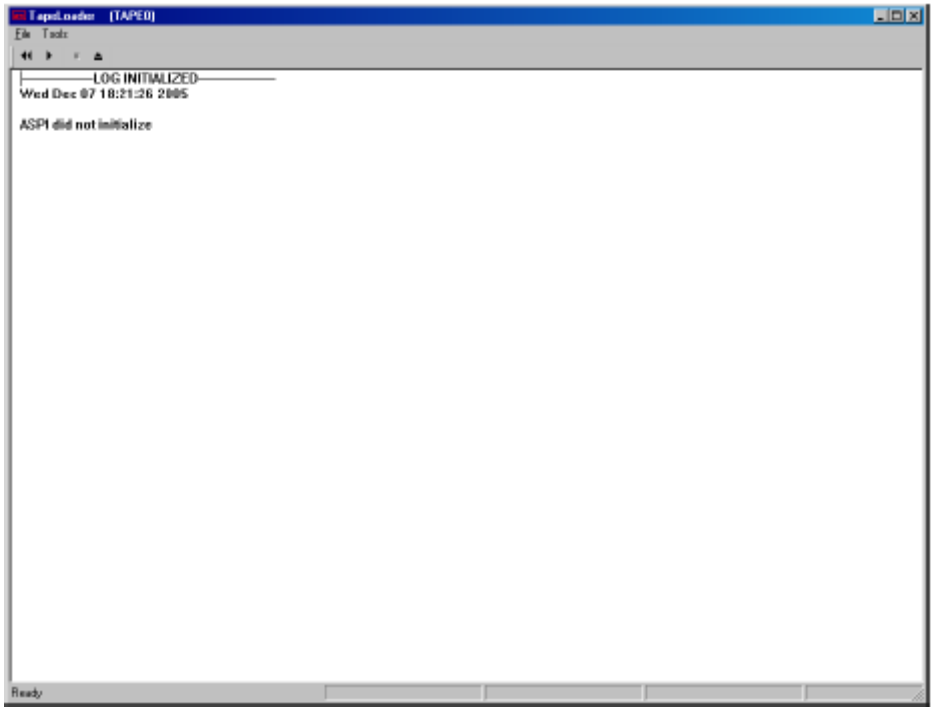
The program is designed for copying the files from the tape to computer disk. The program reads the files of any formats in series as they are and saves them to the indicated directory on the disk under the given names. The program is implemented as an independent application. To run the program, go to the directory where the **RadExPro** Plus package is installed; find the file **TapeLoader.exe** and run it.

System requirements

Program **Tape Loader** works with tape reading equipment, connected through SCSI-interface, under Win2000 and more recent operation systems. ASPI driver is not obligatory, but if it is absent the program cannot determine the type of tape reading equipment linked to computer and will not mark them as TAPE0, TAPE1, etc.

Working with the program

The main window of the program opens to which the log of the operations is written when you run the program:



The log is initialized and if possible the ASPI level is loaded when you run the program; the general

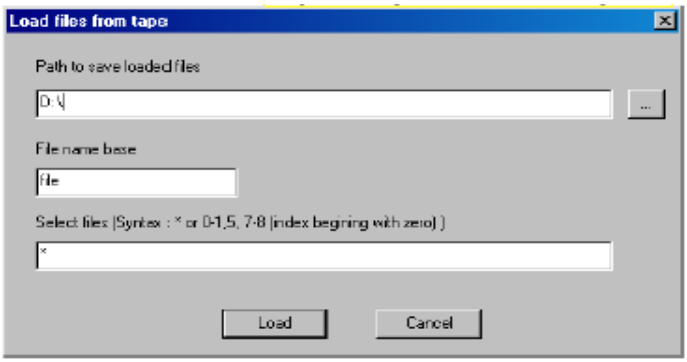
information on initialization process is reflected in the log. Further on the information on the running process is written to the log, along with failures and summary of executed commands.

The main commands, essential for the program, are situated in the menu **Tools**. Buttons on the tools bar duplicates the most part of them.

- The menu **Tools** contains the following commands:

Select tape… - calls the dialog on the current device replacement. (The name of the current device is reflected in the main window header). If the ASPI level is not loaded, the program can't determine the number of linked devices and their names, and uses notation conventions TAPE0, TAPE1 etc.

Load files from tape… - command of running the data loading. As soon as you select the command a dialog of parameter options for saving the loaded files is called.



The parameters have the following meaning:

Path to save loaded files – path to save the loaded files from the tape.

The button **"…"** allows calling the dialog of path selection, as well the path can be selected manually.

File name base – file name base for files on disk to which the number is added (for example, if the file name base = "file", the loaded files will have the names "file0000", "file0001" etc.

Select files – line that sets the file numbers to be loaded from the tape. The following spelling is accepted:

* - read all files from the tape

0, 2, 3 – read the files with indicated numbers (the files' numbering starts from zero, i.e. the first file on the tape is number 0, the second - 1 , etc.)

0-1, 3-5- read the files with the numbers that fall within the indicated range

1, 3-4, 5,7-8 – a combination of separate numbers and ranges is allowed.

After indicating the parameters, click the button **Load** for running loading from the tape and writing them on disk.

- **Rewind** – rewind to the start of the tape.

- **Eject** – ejecting the tape from the device.

- **Break operation** –allows breaking time consuming current operation. There are two types of break operation: the immediate break at the time of reading, or the break after having read the current file. A dialog window appears if you choose this command of break operation; the window allows the user indicating the chosen break mode.

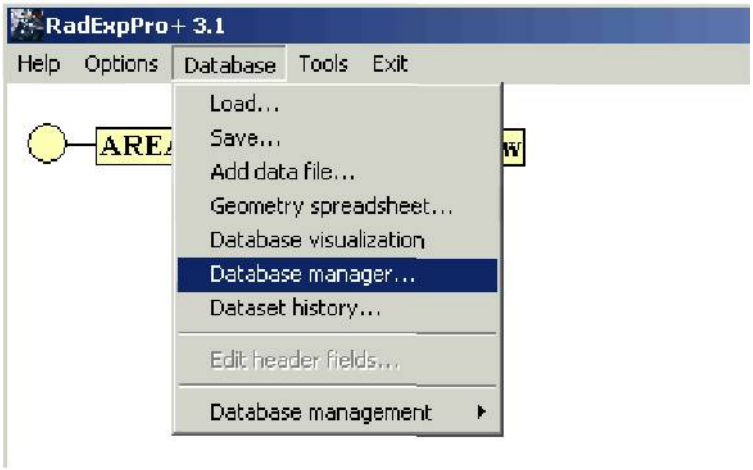
- The **File** menu contains a unique command **Exit** that helps to exit from the program.

Well loading and displaying in the RadExPro package

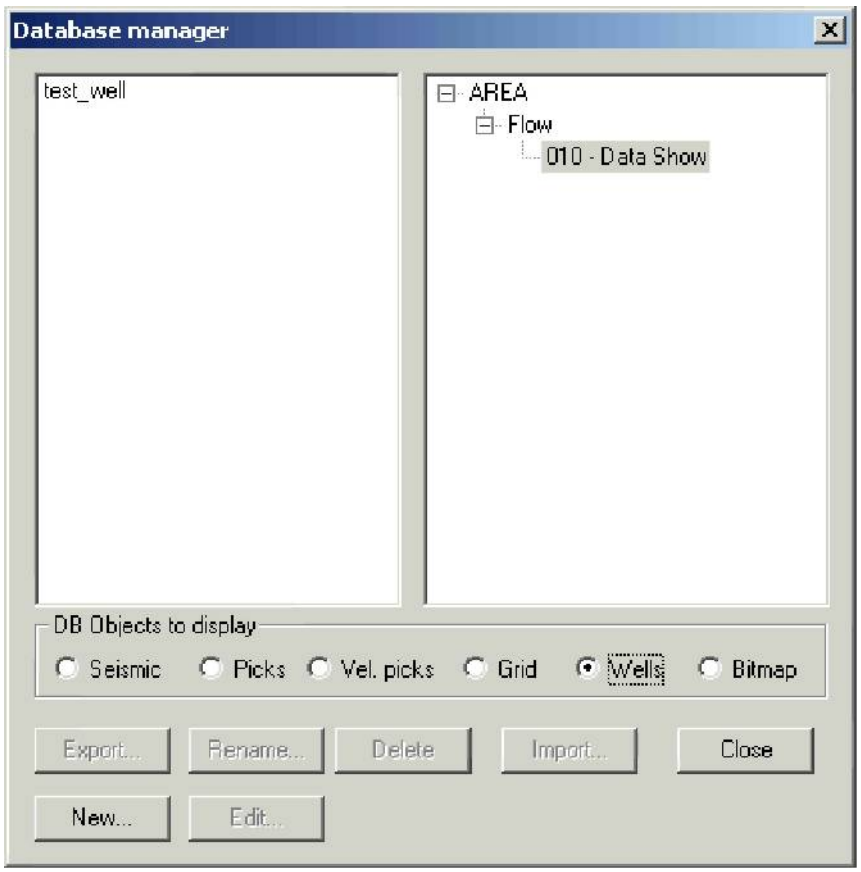
Wells loading to database

The loading of wells into project database is accomplished by the **Database Manager** application.

To open the **Database Manager** window you can use the **Database/Database manager** menu command of the main application window:



In the **Database Manager** window in order to make the operation with database objects - wells - possible, select the **Wells** option in the **DB Objects to display** field:



To load a new well into current database section, click the New... button. The dialog box for well loading will appear. The dialog box consists of three tabs: **Pasport, Data files, Log Data**

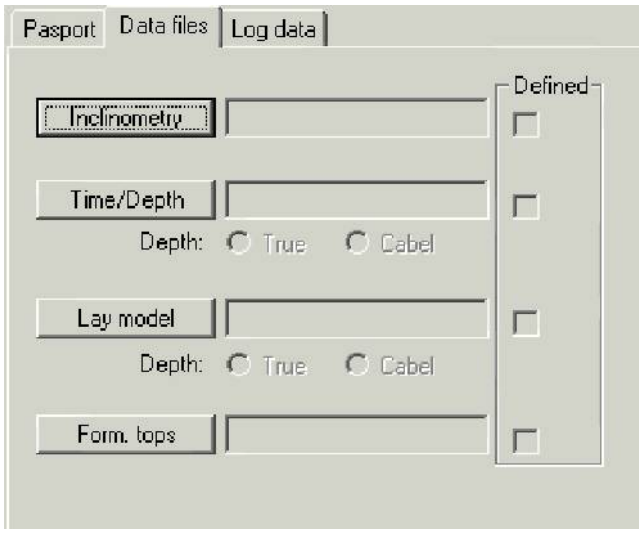
The **Passport** tab is designed to specify general data on well:

The fields in this tab should be filled in by the user. They mean the following:

- **Well title** - the title of the well
- X, Y - relative well coordinates
- L, W - absolute well coordinates
- **Altitude** - wellhead altitude
- **Rotor height** - rotary table height

- **Well depth** - the depth of the well.

The **Data files** tab is designed for loading of data on wells from files:



The following buttons serve to load the following data:

- **Inclinometry** -inclinometry curve obtained in the well.

- **Time/Depth** - time-depth curve obtained in the well.

- **Lay model** - layer model.

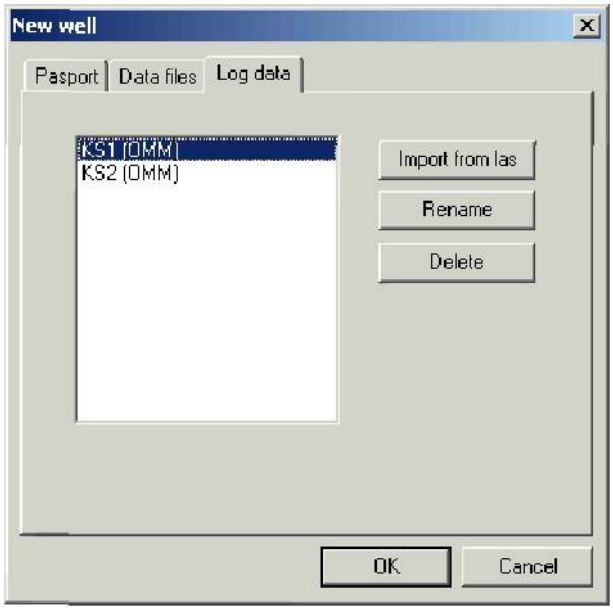
- Form. **tops** - formation tops.

By clicking one of these buttons you can open the dialog box for selection of a file from which the data will be loaded. (see information about file formats in appendix).

When loading the time-depth curve **(Time/Depth)** and layer model **(Lay model),** in the respective **Depth** field, specify the type of depth that should be taken from file: true depth **(True)** or cable depth **(Cabel).**

If the file was loaded successfully then in the field to the right of the button the name of loaded file is displayed and in the **Defined** field which is opposite to the file name the checkmark justifying that the well possesses this type of data is put.

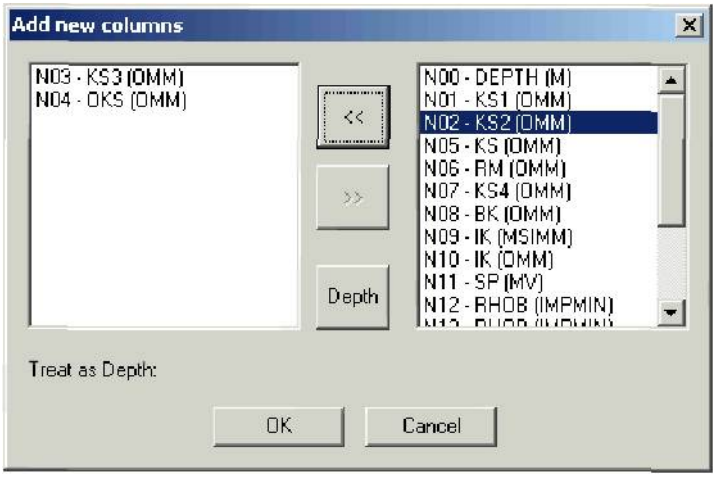
The **Log Data** tab serves to load the well log data.



To the left the list of log curves loaded at the moment is shown. In the brackets next to the name of a certain curve the units of measurement are displayed.

With the help of the **Import from las, Rename** and **Delete** buttons you can import new log curves from files, rename curves and delete the loaded curve, respectively.

To add a new curve to the list, use the **Import from las** button which opens the dialog box for LAS file selection. When selecting a file of supported format the following dialog will appear:



In the right part of the window there is a list of all log curves available in the specified file. In the left part of the window there is a list of log curves that should be added.

By " and " buttons you can choose the log curves that should be loaded to database.

Besides, you indicate in what column the depth values are recorded in the file. To do this, select the required curve (that represents depth) in the list at the left side and then click the **Depth** button. After that, the information about what column to be treated as depth will be displayed in the bottom part of the dialog box in the **Treat as Depth:** field.

After the user has correspondingly filled in all the tabs and pressed the **OK** button the standard dialog box of object saving into database will be displayed. In this dialog box you can adjust the path to database and the name for the new database element.

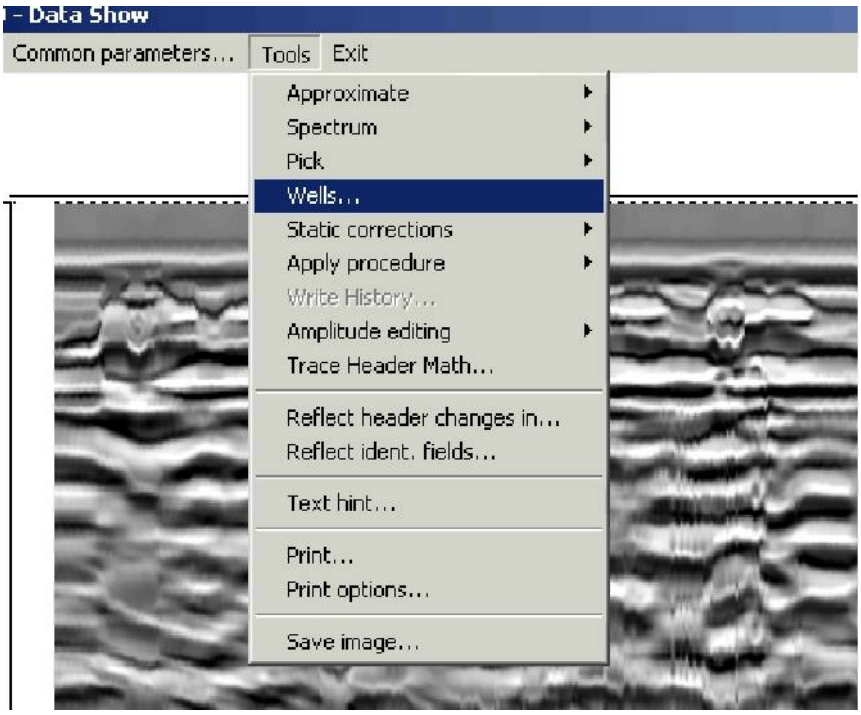
Editing of database elements - wells

If you choose some object - well - in the **Database Manager** window and then presses the **Edit** button, the dialog for well data editing will appear. This dialog box is almost similar to that of new well loading. Via this dialog box you can change the well passport data, various curves and well log data.

Well displaying

The wells are displayed in the **Screen Display** visualization module as projections onto seismic profile.

To select wells that should be displayed and to adjust wells visualization in the *Screen* ***Display*** module, select the **Tools/Wells...** menu command:



When this command is chosen the following dialog box for well visualization parameters settings appears:

In the left part there is a list of displayed curves. To the right of the list there are buttons meant for managing the list.

- **Add well...-** adds a well from the database into the list of displayed curves.

- **Remove well** - removes the selected well from the list of displayed curves.

- **Save well list...** - allows saving the whole list of displayed curves into database in order to load them later.

- **Load well list** - allows loading of previously saved list of wells from database and makes it current. When doing this all wells from the list are being checked.

- **Add well list** - this command is similar to the previous one. However the list loaded from the database does not replace the current one but is added to it.

In the right part of the dialog box there is a set of parameters for well displaying parameters control.

- **Treat vertical axis as Depth** - if this option is activated then the vertical well axis is treated as depth axis. Otherwise, the vertical well axis is treated as time axis.

- **Show well curves** - show/don't show wells.

- **Show formation tops** - show/don't show formation tops.

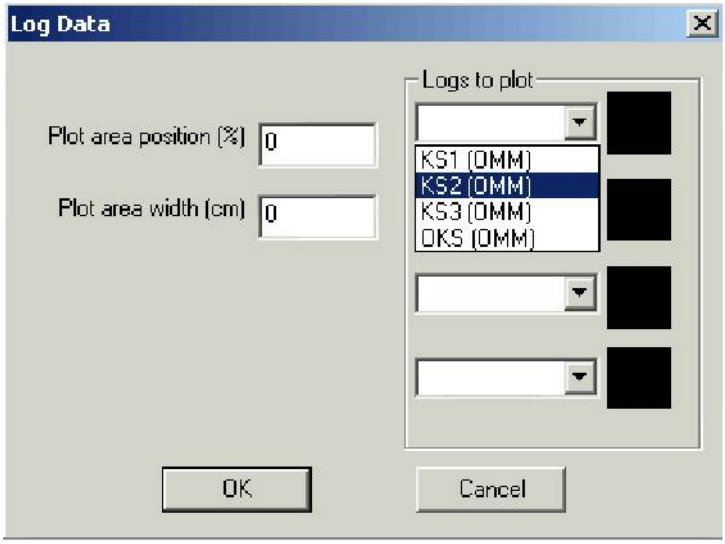
- **Show well names** - show/don't show well name at the top.

- **Stack type** - the stack type on which the wells are projected.

- **Maximal distance to visualize well** - sets the maximal distance from well to profile at which the well point is still projecting onto profile and reflecting.

- **Use time-depth curve if available** - makes application of time-depth curve obtained from the well a priority above the application of time-depth curve obtained from **Velocity Model Editor.**

The **Log Data...** button should be used to adjust the parameters for log curves displaying for every selected well. When this button is pressed the following dialog box appears:



It is possible to display up to four log curves for every well at the same time. In the right part of the dialog box in the **Logs to plot** field there are four drop-down lists of available for this well log curves. Here you can select the curves which should be displayed on the screen. For every curve you can specify the color by clicking the colored square to the right.

At the left side you can specify uniform display parameters for all selected log curves for this well:

- **Plot area position** - the displacement of log curve zero from the borehole. It should be expressed as percentage from maximal amplitude.

- **Plot area width** - the width of the area covered by the maximal curve amplitude. Should be expressed in centimeters.

File formats

Formation top file description

Every line in the file is of the following kind:

<depth> <abbreviation> [<full name>]

(double) (contains up to 10 symb.) (contains up to 235 symb., optionally)

Besides these lines the following lines are possible

- The lines staring with '#' symbols (comments),

- The lines staring with '\n' symbol (carriage return) The presence of lines of other formants is prohibited.

A file description

The file must contain the line starting with ~A. In this line the columns names of successive table are described.

All lines following this line must be the table rows with blanks or tabs as separators. The presence of insignificant lines started with '#' or '\n' symbols is permissible.

Inclinometry file description

File of ~A type (see above).

Must contain compulsory buttons: Depth, X, Y, Z.

Other columns are permissible. Must contain at least one record.

Time/Depth file description

File of ~A type (see above).

Must contain the following columns: Time and Depth (if the depth is cable one) or Time nad Z

(if the depth is true one).

Refraction mode

WARNING!: This part of the software is obsolete is kept for backward compatibility only. It is not recommended that you use the Refraction Mode. For refraction data processing use *Easy Refraction* and *Travel Time Inversion* modules instead.

The Refraction mode is switched on and off using the **Options/Refraction mode** entry of the main menu of the project tree window.

Note that when the Rafraction mode is on, the response to the mouse click of database objects of the levels “Area” and “Profile” of the project main window changes. The double click on the left mouse button on the “Area” object leads to the opening of the specialized profiles’ location of the Correlation Refraction Seismic data interpretation, not a Map.

A detailed characteristic of the obsolete Refraction mode has been removed from the manual. If you still need it for any reason, please contact the RadExPro support team.

.

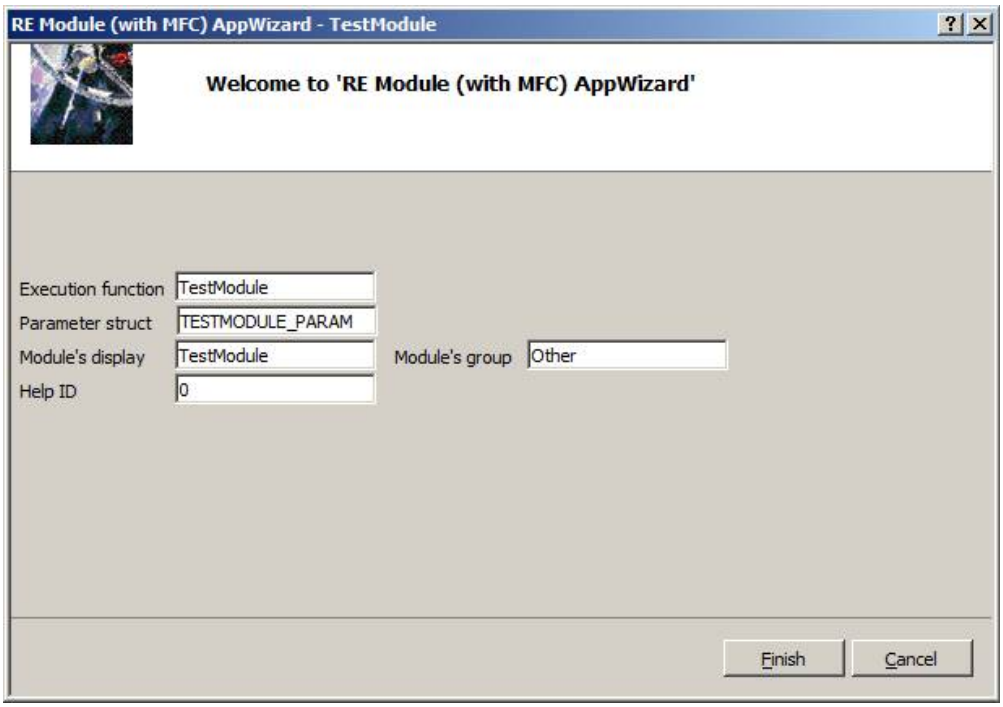
Development tools - Creation of your own module

Microsoft Visual C++ Custom AppWizard

Here we provide just a brief overview of how to create a *RadExPro* module. If you actually like to program your own module, please contact us for instructions, as well as necessary libraries, include files, etc.

To create a template for a **RadExPro** module within ***Microsoft Visual* C++ *6.0*** or ***Microsoft Visual* C++ 2005** environment, a special AppWizard was developed. Please, contact us for the required files and installation instructions.

After installing the Wizard, open MS Visual Studio and create a new Visual C++ project. Select *RE Module* project type. This will start the RadExPro Wizard, the following window will open (for MS VC++ 2005 Wizard):



The parameters here are as following:

- **Execution function** - the name of the main function of the module which will be exported from dll and executed by the host processing flow.

- **Parameter struct** - the name of the structure with module parameters.

- **Module's display** - the name of the module as it will appear in the list the RadExPro modules in the flow editor.

- **Module's group** - the name of the group to which the module will be assigned.

- **Help ID** - the index in help file (currently unused).

The Wizard creates the empty frame for a RadExPro module, all exported functions required, and some auxiliary internal functions as well as empty parameter structure. Now a developer shall fill in the functions with the code.

FUNCTIONS

int __stdcall INITPARAMS(HWND own_wnd,

int *buf_len, TEST_MODULE_PARAM *par)

The function initializes module parameters structure (in our case, of TEST_MODULE_PARAM type).

*buf_len contains the size of the parameter structure.

When the module is added to the flow for the first time, the *buf_len is equal to -1. Then the internal function InitDefaultParameters is called and initializes the parameters structure with the default values.

If *buf_len is not -1, it is checked if the size correctly corresponds to the actual structure size. If positive, the internal function EditPaarmeters is called. This function shall normally show the module parameters dialog window.

int InitDefaultParameters(TEST_MODULE_PARAM *par, EXTRA_PARAM *ep)

You must implement this function to set the default parameters of the module.

int EditParameters(HWND w,

TEST_MODULE_PARAM *par, EXTRA_PARAM *ep)

Implement this function to call module parameters dialog and allow user to edit module parameters. The updated values shall be saved to *par.

void __stdcall ExtraParameters(EXTRA_PARAM *ep)

It is recommended not to do anything with this function. It is simply writes down the

EXTRA_PARAM pointer into global module variable and locks the critical section.

Main module function:

unsigned long stdcall test_module(COMMON_PARAM *cp, TEST_MODULE_PARAM *par)

(The name of the function depends on how you called it in the Wizard dialog).

This function is called by the flow being executed. Here the module algorithm is to be implemented. It receives a pointer to COMMON_PARAM structure which contains seismic data (current frame), auxilary information and means of access to trace headers information, and a structure with the module parameters.

The definition of the COMMON_PARAM is shown below. The most important parameters and funtions that you would likely need to use are indicated in the comments:

```
typedef struct___common_param

{

        unsigned long state;

        int n_tr; //Number of seismic traces in the frame

        STRACE * tr; //Array of seismic traces. Typically you would access the i-th trace sample

// of the j-th trace as cp->tr[j].d[i]

        double dx, dt; //dt is sample interval of the traces in the flow in ms

        int np; // Number of samples per trace in all traces of the flow

        void far _export (far*DepictWorkPercent)(int Percent);//pointer to function that displays
                //percentage of function completion.
```

```
    int p2n_tr;
    HWND MainWnd; // handle to main program window
    GENERAL_OPTIONS gopt;
    float x_start, t_start;
    DM_DATAHANDLE dh;
    float y_start;
    float dy, dz;
    char l_type;
    float v;
    int nx, ny;
    int has_headers;
    int nf;
    DMP_DATAELEMDEF *de;
    DMP_TRACEHEADERS th;
    int *rsort_ind;
    int (*GetHNameInd)(struct __common_param *cp, char name[]); //pointer to a function that
// returns index of a header with the given name. First argument (*cp) shall be a pointer to
```

```
// COMMON_PARAMS structure, typically to the host COMMON_PARAMS.

        double (*GetHField)(struct __common_param *cp, int ind, int itr); //pointer to a function
that

// returns a value of ind  header fiedl of the itr trace

        void (*SetHField)(struct __common_param *cp, int ind, int itr, double v); //pointer to a

//function that sets a new value to ind  header fiedl of the itr trace.

//Typically you would access a trace header field the following way:

//

//   int indxSOU_X = cp->GetHNameInd(cp, "SOU_X");

//  if ( indxSOU_X > -1)

//     {

//       for (int i=0; i<cp->n_tr; i++) //for all traces

//             {

//               double valueSOU_X = cp->GetHField(cp,  indxSOU_X, i); //get the SOU_X value

//                  ... //do something with the value here, e.g.  modify it

//               cp->SetHField(cp,  indxSOU_X, i, valueSOU_X); //save modified value back to header


//             }
```

```
//      }

//

        void far _export (far*DepictStatusStr)(char s[]);//send a status string to the flow status winow

        int (*GetNextEnsemble)(struct___common_param *cp, int l, int *r);//gets indexes of the first //(*l) and last (*r) traces of the next ensemble in the current frame. Ensemble is defined by the first

// sorting key of  the Trace Input module.

        int nsf_ind;

        int sf_ind[10];

        int cuse_s, cuse_n;

        int cfr_ind;

        int tot_ntr;

        void *mc;

        void (*SetModuleData)(unsigned data);

        unsigned (*CreateDataOutput)(struct ___common_param *cp,

        unsigned type, unsigned s_class); int

        (*TraceOutput)(unsigned stream, OUTTRACE *tr);

        char flow_title[128];
```

```
        IStream **crdconn;

        IREExecConn *exconn;

}

COMMON PARAM;
```